Section 10
Nonpoint Source Model

Purpose

The BASINS *Nonpoint Source Model (NPSM)* is a planning-level watershed model that integrates both point and nonpoint sources. It is capable of simulating nonpoint source runoff and associated pollutant loadings, accounting for point source discharges, and performing flow and water quality routing through stream reaches and well-mixed reservoirs. *NPSM* uses most of the simulation capabilities of the Hydrologic Simulation Program - FORTRAN (HSPF). For a detailed description of the simulation algorithms used in the HSPF, refer to *Hydrological Simulation Program - FORTRAN, User's Manual for Release 11.0 (Bicknell et al., 1996)*, published by the U.S. Environmental Protection Agency, Environmental Research Laboratory, Athens, Georgia. That manual can also be downloaded from the BASINS Web site.

Application

NPSM provides a graphical user interface (GUI) that can be launched directly from the BASINS View or as a stand-alone program. The model can be used to simulate a single watershed or a system of multiple hydrologically connected watersheds. Each watershed is defined as a hydrologic unit containing a series of point and nonpoint sources discharging to a unique stream reach. When using *NPSM* within the BASINS GIS environment, a watershed can be subdivided into a number of segments (or subwatersheds). Subwatershed boundaries can be either developed through an on-screen delineation process using the BASINS *Watershed Delineation* tool or imported using the BASINS *Import* tool. Imported watersheds are generally watershed boundaries that have been delineated and digitized. When *NPSM* is executed from the BASINS View, the corresponding land use distribution data, stream characteristics data, and point source data are extracted and prepared for integration into the model's interface. *NPSM* interface requires the designation of meteorological data and the simulation period, as well as the model parameter values required to build the model input file and ensure a successful simulation.

NPSM can be applied to support various watershed and water quality modeling studies. Examples of such studies include the following:

- Simulation of watershed existing conditions and evaluation of the current water and water quality status.

- Simulation of future land use change effects on both the water balance and water quality loading.

- Simulation of various point and nonpoint source control strategies.

- Development of watershed or subwatershed controls necessary to meet specific water quality goals.

Procedures

> ## *Key Procedures*
>
> ✓ *Activate and select the watershed theme for modeling*
>
> ✓ *Select NPSM from the Models menu*
>
> ✓ *Select NPSM options in the "Non-point Source Model Options" window*
>
> ✓ *View and/or modify the default perviousness percentages associated with land use categories*
>
> ✓ *Proceed through the NPSM interface, making necessary settings*
>
> ✓ *Run NPSM*
>
> ✓ *View NPSM output in the NPSM postprocessor*

10.1 *NPSM* Execution from the BASINS View

1. While in the BASINS View, display and activate the theme containing the boundaries of the watershed(s) to be modeled by checking the box next to the theme name and clicking on the theme title.

2. Select the watershed(s) by clicking the **Select Feature** button and creating a small box within the watershed(s) using the left mouse button. When modeling a multiple-subwatershed system, ensure that the watersheds are hydrologically connected.

Tip: *Use the SHIFT key on your keyboard and the **Select Feature** tool simultaneously to select multiple watersheds if all watersheds cannot be selected at once by creating a single box.*

Tip: *Each subwatershed must contain at least one unique stream reach (either Reach File, Version 1 or Reach File, Version 3), and watershed boundaries must have been delineated using the BASINS **Watershed Delineation** tool or imported using the BASINS **Import** tool (selecting BASINS Watershed as the import file type). If a multiple-subwatershed system is being modeled, the watersheds must be connected by reaches.*

---

3. Once the simulation domain is defined (watershed(s) is selected), *NPSM* can be launched by selecting *NPSM* from the *Model* pull-down menu.

4. Select NPSM options in the "Non-point Source Model Options" window (Screen 10.1.1). First, enter a project name for simulation of the selected watershed(s); use eight characters or fewer for the project name and do not include a file name extension. A subdirectory denoted by the project name is created within the BASINS\MODELOUT directory at the root of the drive selected for BASINS installation. The subdirectory will contain all BASINS-created files required to run *NPSM* for the selected watershed(s). Next, select the discharge year for Permit Compliance System (PCS) data to be incorporated into the model. A file containing average flow and loading values for the discharge year selected will be created for each point source facility located within the watershed(s) being modeled. Finally, select whether to modify the default percent perviousness value associated with each land use category. In most situations this is recommended. Checking the box will enable you to view and update/edit the default perviousness percentages for each land use category represented in the model. Click **OK** to proceed or **Cancel** to end execution of *NPSM*.

Screen 10.1.1

5. Select the appropriate land use representation for modeling (Screen 10.1.2). You will be prompted to select the appropriate land use representation only if you have imported a new land use coverage into BASINS by using the BASINS *Import* tool (and have selected BASINS Landuse as the file type) or if you have reclassified a land use theme using the BASINS *Landuse Reclassification* tool. In all other situations, the USGS Land Use distribution is used.
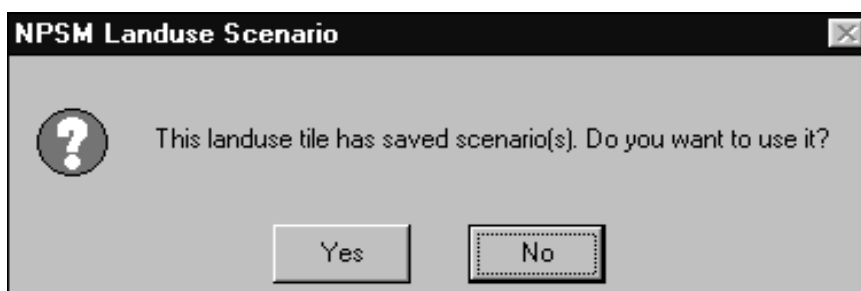

Screen 10.1.2

*Tip:* *If a local or updated land use data layer is available and you opt to use it for modeling, this layer should be added to the BASINS theme using the BASINS **Import** tool. If a land use data layer is imported/added as a standard import, it will not be recognized by **NPSM**.*

6. If a land use scenario(s) was developed using the BASINS *Land Use Reclassification* tool, a dialog box will provide an option to select a land use scenario (Screen 10.1.3). Click **Yes** to choose a scenario. Click **No** to use the selected land use theme's original classification.

Screen 10.1.3

If you choose to select a land use scenario, navigate to the directory containing the land use scenario file disignated with a SCN file extension (Screen 10.1.4). Choose a file and Click **OK**. The next dialog provides an option to view the scenario attributes including name, creator, and date. You can accept the selected scenario or return to Screen 10.1.4 to choose a different scenario file.
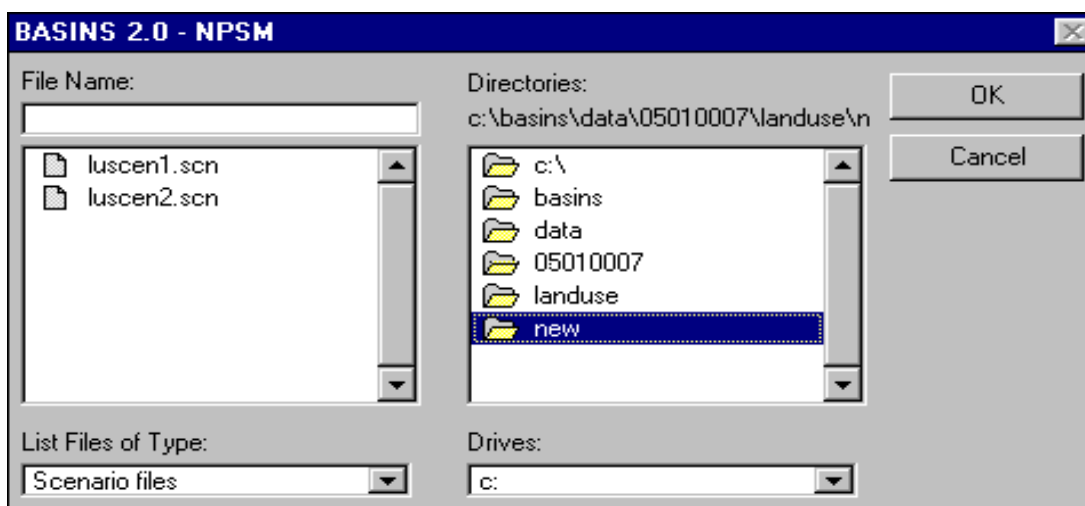
Screen 10.1.4

7. View and modify the default perviousness percentages associated with the land use categories represented for your simulation in the "Modify Percent Perviousness" window (Screen 10.1.5). The default land use representation for modeling is a lumped Anderson Level II representation consisting of the land use groupings below (individual land use codes that have been lumped are in parentheses). These land use categories are located in the "Select a Landuse Category" list. However, if a new land use coverage was imported into BASINS by using the BASINS *Import* tool (and selecting BASINS Landuse as the file type) or a land use theme was reclassified using the BASINS *Landuse Reclassification* tool, different land uses will appear in the list. Land use categories in these situations will consist of those designated during import or reclassification. No automated grouping of land uses will be performed (with the exception of any groupings made when using the BASINS *Landuse Reclassification* tool).

Urban or Built-up Land	(10-19)
Agriculture Land	(20-29)
Rangeland	(30-39)
Forest	(40-49)
Water	(50-59)
Wetland	(60-69)
Tundra	(80-89)
Perennial Snow or Ice	(90-99)
Barren Land	(70-79)
Unclassified	(0)

NPSM extracts land use areas for each watershed being modeled and divides them into separate pervious and impervious land units for modeling. Therefore, each land use must be assigned a perviousness percentage. When a land use is highlighted in the "Select a Landuse Category" list, the corresponding default perviousness percentage is visible in the "Enter percent perviousness" box. This number, which can be edited, must be an integer between 0 and 100. The percent perviousness value is used directly to subdivide each land use category into pervious and impervious land units. For example, if a percent perviousness of 25 is assigned to Urban or Built-up Land in a watershed containing 500 acres of Urban or Built-up Land, two separate land units will be identified in the model—a pervious Urban or Built-up Land unit of 125 acres and an impervious Urban or Built-up Land unit of 375 acres.
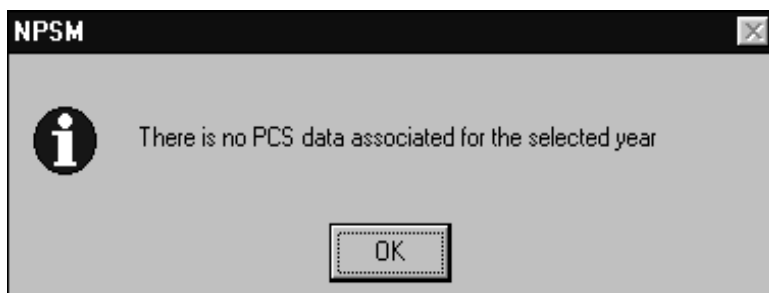

Screen 10.1.5

> *Tip:* If the default percent perviousness values for land use categories are changed, the changes can be made to the default set by checking the "Save the changes" box. If the changes made are intended only for the present simulation, the "Save the changes" box should not be checked.

Click **OK** to proceed or **Cancel** to halt execution of *NPSM*.

> *Tip:* Each land use category is subdivided by **NPSM** into two land units, based on the perviousness percentages. The first land unit represents the pervious portion, and the second land unit represents the impervious portion. Note that the model uses different simulation algorithms to represent the runoff and pollutant loading processes associated with pervious land and impervious land.

> *Tip:* You can expect the data extraction and preparation for **NPSM**, following the definition of land use perviousness, to take a few minutes, depending on the size and location of the study area, the number of subwatersheds to be modeled, and the characteristics of the hardware used.

If no point source data are available for the watershed(s) you are modeling, a warning window appears (Screen 10.1.6). Simply click **OK** to continue. In this situation, no point source discharge data are passed to the model; however, flow and concentration values for any pollutant and any point source facility can later be entered directly into the *NPSM* interface (see Section 10.9).
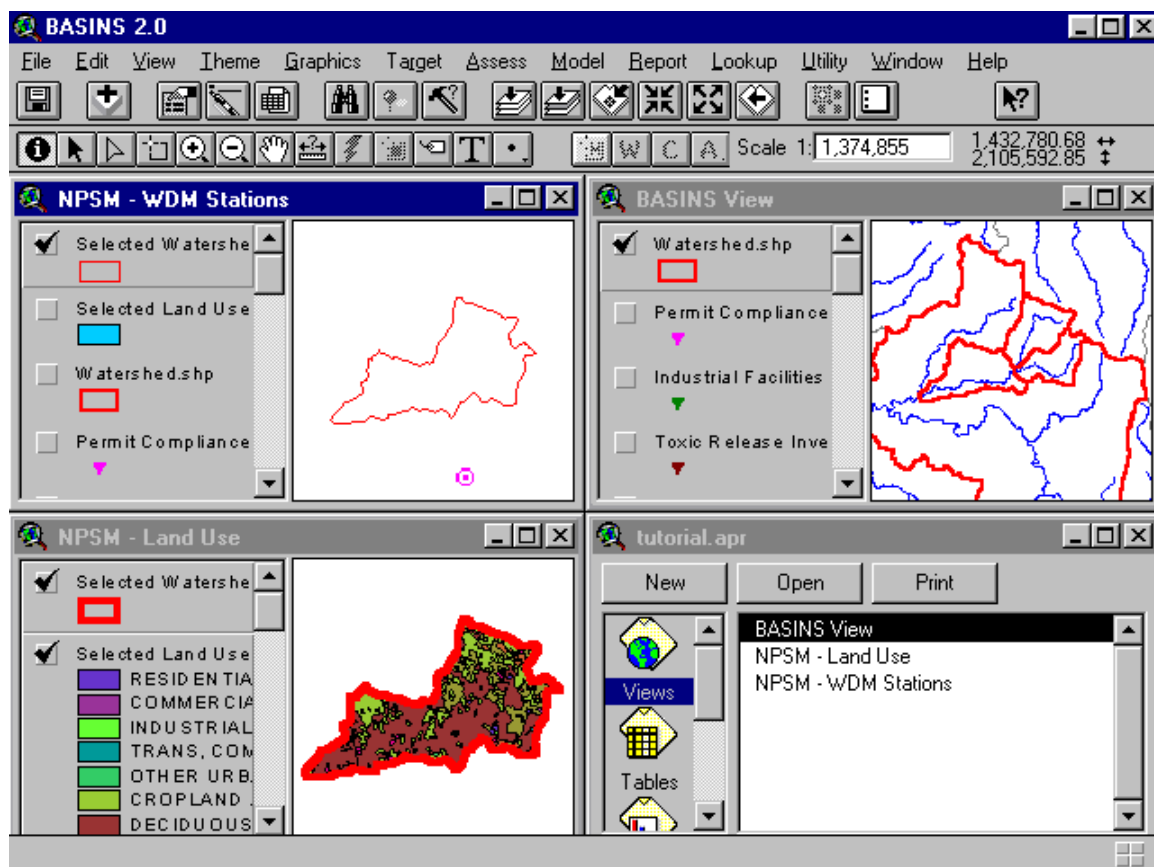

Screen 10.1.6

When *NPSM* is launched, the BASINS GUI appears in the background of the "Nonpoint Source Model" window, referred to as the *NPSM* interface. The BASINS GUI displays four graphical windows (Screen 10.1.7), which contain the following information:

- Watershed Data Management (WDM) weather stations in close proximity to the selected watershed(s).

- Land use distribution for the selected watershed(s).

- BASINS View.

- BASINS project window.

Screen 10.1.7

Tip: The BASINS GUI can be viewed by minimizing the *NPSM* interface or clicking the cursor on any portion of the BASINS GUI.

8. The *NPSM* interface (Screen 10.1.8) is a window containing a series of 14 interface buttons located immediately below a menu bar. The first two buttons from the left should be the only active buttons:
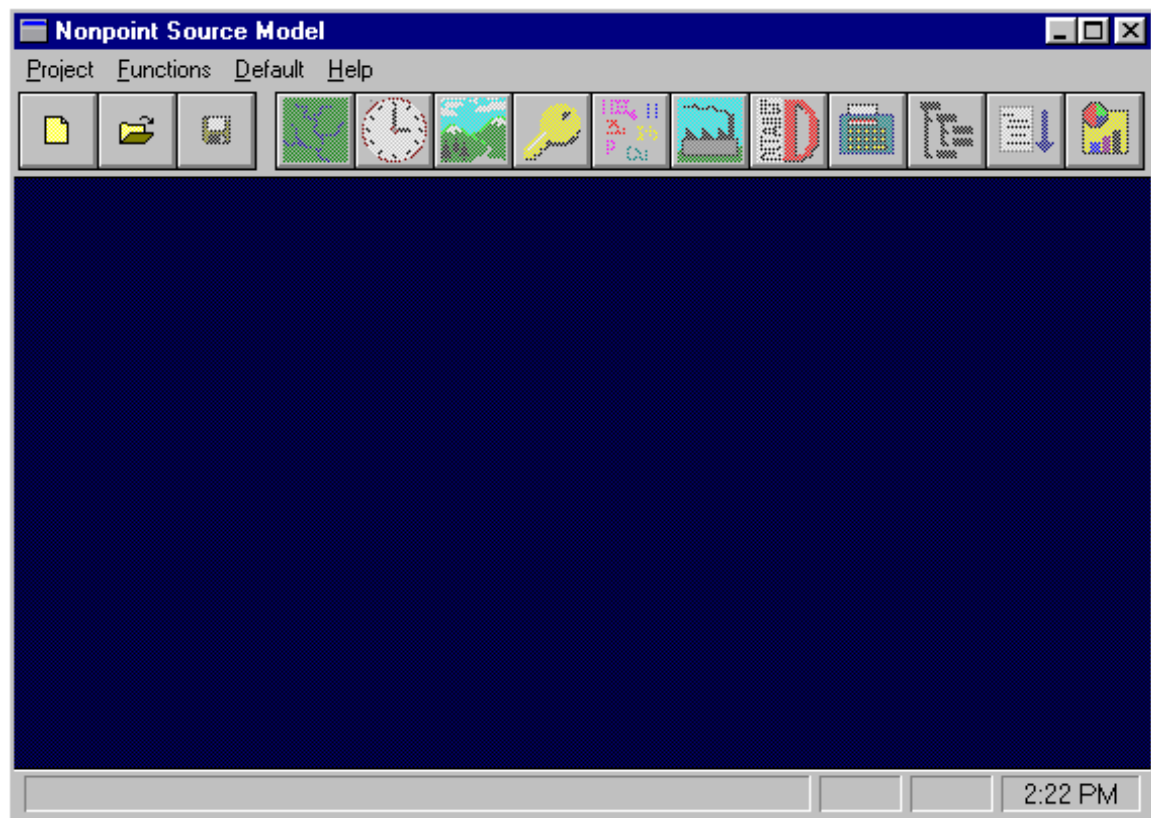
Create a new project

Open an existing project

Tip: *Increase the **NPSM** interface window size to view all interface buttons.*
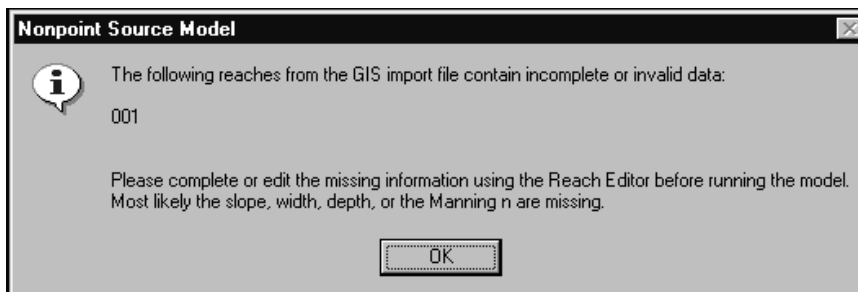

The remaining buttons, which do not become active until a new project is created or an existing project is opened, are described later in this section. Also note that the BASINS GUI remains in the background of the *NPSM* interface. The GUI, along with any of its tools or functions, can be accessed during activity in the *NPSM* interface simply by clicking on it. For example, you can (1) perform a characterization of the soil in the watershed(s) being modeled to estimate the soil permeability, (2) assess the distribution of elevations in each subwatershed, (3) review the distribution of point sources and their annual loading for specific pollutants for a given year, or (4) review available water quality data.

Nonpoint Source Model

Project   Functions   Default   Help
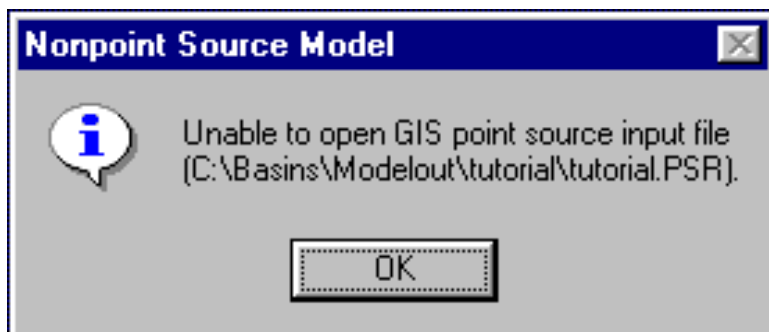
Screen 10.1.8

9. The GIS data extracted for the current study area are sent directly to the *NPSM* interface, where a new *NPSM* project is created.

    If *NPSM* is executed on a Reach File, Version 3-level watershed or if Reach File Version 1 data are incomplete for the reach contained in the watershed being modeled, a warning window appears (Screen 10.1.9). Click **OK** to continue. In this situation, reach data passed to the model are incomplete and must be edited within the Reach Editor.

Nonpoint Source Model

The following reaches from the GIS import file contain incomplete or invalid data:

001

Please complete or edit the missing information using the Reach Editor before running the model. Most likely the slope, width, depth, or the Manning n are missing.

OK

Screen10.1.9

Once again, if no point source data are available for the watershed(s) you are modeling, a warning window appears (Screen 10.1.10). Simply click **OK** to continue. In this situation, no point source discharge data are passed to the model; however, flow and concentration values for any pollutant and any point source facility can later be entered directly into the *NPSM* interface.

Screen 10.1.10

You will be prompted to save this new *NPSM* project. Enter a project name (use eight characters or fewer to create the project file; a .prj extension is automatically assigned by the system) (Screen 10.1.11). Click **Save**. This project will be saved in the BASINS\MODELOUT\ directory within the subdirectory associated with the watershed(s) you are modeling.
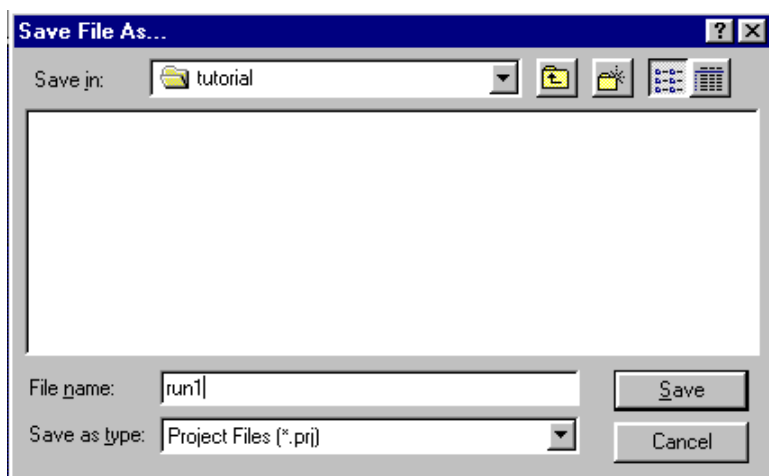
Screen 10.1.11

Notice that all of the *NPSM* buttons are now active and in full color.

TUTORIAL

- *Display and activate watershed.shp.*

- *Select all three subwatersheds of watershed.shp.*

- *Name your project <u>tutorial</u>.*

- *Select 1993 as the PCS discharge year. Check the box to view and update/edit the default percent perviousness.*

- *Assign a perviousness percentage of 100 to any Agriculture Land, Rangeland, Forest, or Barren Land and a perviousness percentage of 50 to Unclassified or Urban or Built-up Land. Do not check the "Save the Changes" box.*

- *After the **NPSM** interface appears, save the project as <u>run1.prj</u> in the BASINS\MODELOUT\TUTORIAL directory.*

10.2 Executing NPSM as a Stand-Alone Program

The *Nonpoint Source Model (NPSM)* can also be executed as a stand-alone program. An existing project can be opened, a new project can be created using data previously extracted during NPSM execution from the BASINS View, or a new project can be developed from scratch. No tutorial steps are provided in this section. These steps assume that you are already familiar with executing NPSM from BASINS and specifying model inputs.

1. From the Windows Taskbar at the bottom of your computer screen, select the Windows **Start** button.

2. Select *Programs* from the *Start* menu.

3. From the *Programs* menu, select *BASINS*.

4. Select *Nonpoint Source Model* from the *BASINS* menu.

5. The NPSM interface appears in much the same manner it does when executing NPSM from the BASINS View. The only exception is that the BASINS GUI is not present in the background of the NPSM interface.

6. At this point, a new NPSM project can be created using previously extracted data, an existing NPSM project can be opened, or a new NPSM project can be created from scratch.

    a. To create a new project using data extracted previously by executing NPSM from the BASINS View, select *New* from the *Project* heading or click on the **Create a new project** button.

    The following window prompts you to "Select Landuse File". Be sure that the directory in the "Look in:" box refers to the watershed(s) you wish to model. This directory should be located within the BASINS\MODELOUT\ directory. A file denoted by your project name and a .wsd extension should appear in the file list. Highlight this file and click **Open**.

    A warning window appears if the previously extracted data represents a Reach File, Version 3 segment or a Reach File, Version 1 segment with incomplete data. Click **OK** to continue.

    If no point source data are available for the watershed(s) you are modeling, a warning window will appear. Simply click **OK** to continue.

    You will be prompted to save the project being created. Enter a project name (use eight characters or fewer; a .prj extension is automatically assigned). This project will be saved in the BASINS\MODELOUT\ directory within the subdirectory associated with the watershed(s) you are modeling.

    b. To open an existing project or to create a new NPSM project from scratch, select *Open* from the *Project* heading or click the **Open an existing project** button.

    The window that appears prompts you to "Select Project File...". If you are opening an existing project, be sure that the directory in the "Look in:" box refers to the watershed(s) you previously modeled. This directory should be located within the BASINS\MODELOUT directory. A list of

NPSM project files should appear in the file list. NPSM project files are denoted by a .prj extension. Highlight the file you wish to open and click **Open**.

If you are creating a new NPSM project from scratch, locate and open the SAMPLE.PRJ file located in the BASINS\MODELOUT\SAMPLE directory.

Notice that all of the NPSM buttons are now active and in full color.

10.3 NPSM Interface

The NPSM Interface (Screen 10.3.1) consists of the menu headings *Project*, *Functions*, *Default*, and *Help* (all located across the top of the screen), NPSM functional buttons (located directly below the menu headings), and a status bar and clock (located at the bottom of the screen).

Tip: If you move the cursor on top of any button, you will see a brief description of the button function in the status bar.

The NPSM functional buttons alone can be used to complete a successful NPSM simulation. With the exception of the *Default* heading, options listed in the menu headings simply offer another method of performing the NPSM functional button actions. Options for opening and creating a default file are available only from the menu heading.

It is recommended that you proceed through the NPSM functional buttons in order, from left to right. If data are edited in a button located to the left of the screen, it is necessary to make changes to data in every button located to the right of this button. The details and requirements for each button are presented in Sections 10.4 through 10.14.
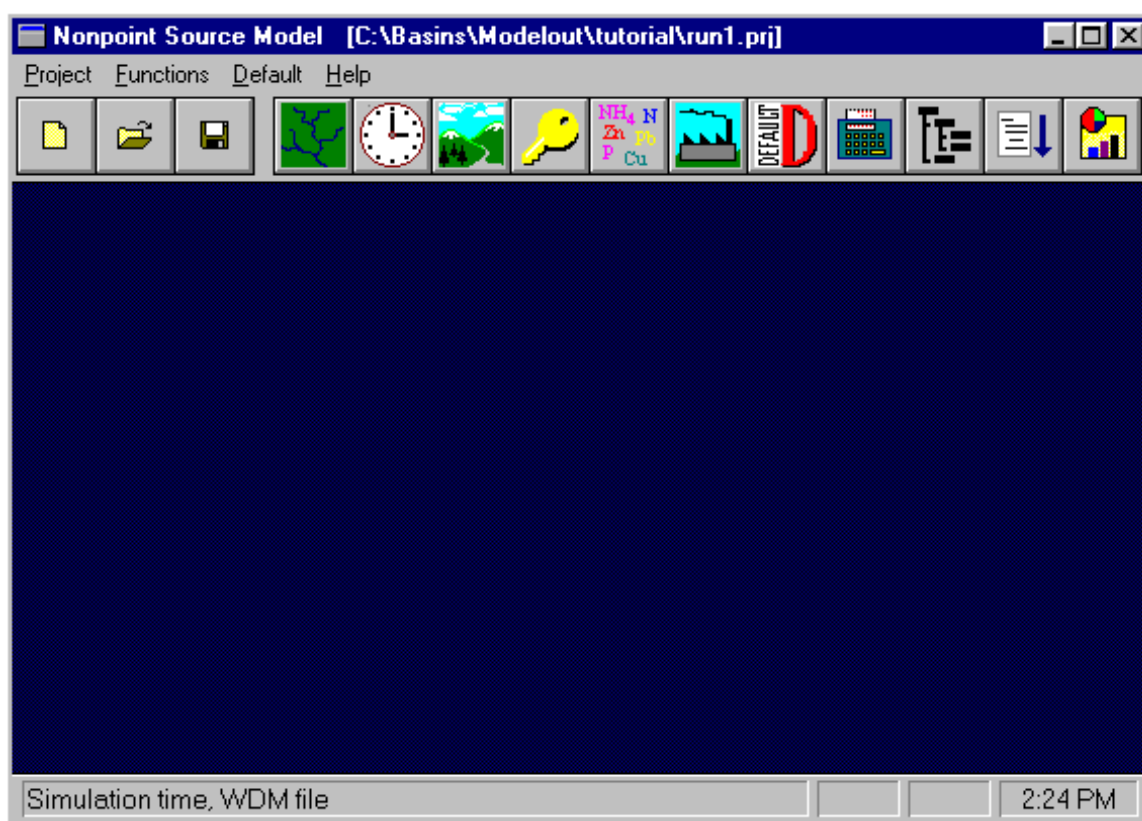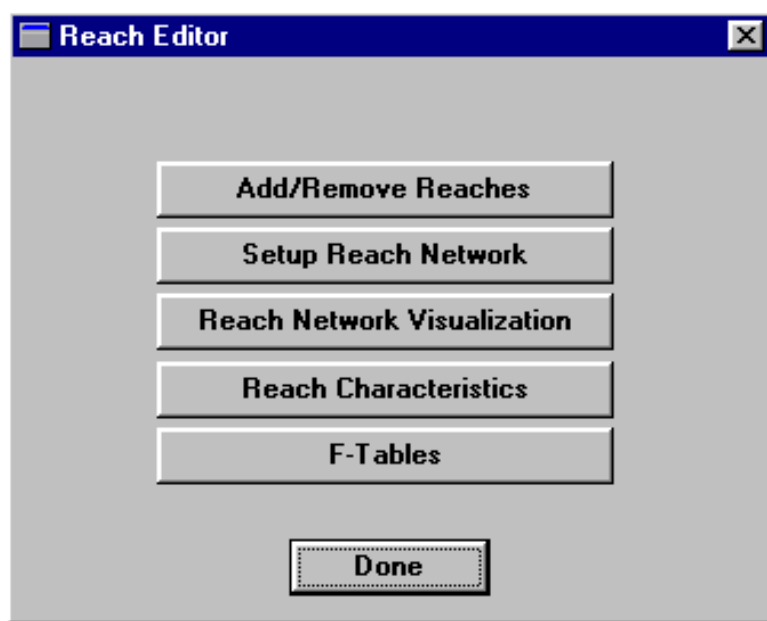

Screen 10.3.1

10.4 Reach Editor

1. Click on this button to view or modify stream reach settings and associated parameters through the "Reach Editor" window (Screen 10.4.1). This window contains five options—*Add/Remove Reaches, Setup Reach Network, Reach Network Visualization, Reach Characteristics,* and *F-Tables*.

Screen 10.4.1

2. Select **Add/Remove Reaches** to view the reaches associated with the watershed(s) selected for simulation. This window (Screen 10.4.2) displays Reach #, Reach Name, Reach ID, # of Exits, Type (Stream or Lake), and Watershed for each reach.

    Be sure that all reaches are designated as Streams because NPSM currently does not simulate lakes.

    Be sure that designations in other portions reflect any changes made to existing reaches in this window.

    The # of Exits should be set to 1, because multiple exits are not currently supported in NPSM.

Tip: When editing cells in this window or any similar window in the NPSM interface, be sure to press the ENTER button on your keyboard to confirm the changes before clicking OK or Cancel.

Tip: Reaches can be added or removed from this screen by clicking on the right mouse button and selecting "Add Reach" or "Remove Reach".

Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes. If you select **OK**, you will be prompted to clear the Output Manager data (screen 10.4.3). Select **Yes** if you have modified the watershed ID number or **No** if you have not.

Screen 10.4.2

Screen 10.4.3

3. Select **Setup Reach Network** to view the reach network in a text format. This window (Screen 10.4.4) displays the Reach #, Reach Name, Headwater, Upstream Left, Upstream Right, Complementary, and Downstream reach for each reach being simulated. This network is set up automatically based on the characteristics derived from the information in the BASINS Reach File, Version 1 and Reach File, Version 3 files.

    Each reach appearing in the "Add/Remove Reaches" window should also be present in this window. Any reach that is a headwater should be designated with a "Yes" in the Headwater column. If you need to change the headwater designation, click on the cell you wish to change and click on one of the two radio buttons at the top of the window (**Yes** and **No**).

    The Upstream Left, Upstream Right, Complementary (reach feeding into the most downstream end of the reach in question), and Downstream columns contain the appropriate reach numbers of the surrounding reaches. A value of -999999 for any of these categories (which is acceptable in many situations) signifies that no surrounding reach matches this identification. Headwater reaches should always have values of -999999 for the Upstream Left and Upstream Right categories.

    Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.4.4

4. Select the **Reach Network Visualization** button to view the reach network graphically. The "Reach Graph" window (Screen 10.4.5) displays information from the "Setup Reach Network" screen in a graphical manner. It also allows you to add, remove, and move reaches in a visual manner.

    Each blue line segment represents a separate reach. Moving the cursor onto a segment changes the segment color to green and displays a box containing reach identification information. The

identification information includes Reach name, Reach #, Watershed ID, Reach ID, ULCSM (upstream left), URCSM (upstream right), DSCSM (downstream), and CCSM (complementary).

When the watershed delineation is based on Reach File, Version 3 streams IDs are automatically assigned by BASINS. In the visualization screen the assigned ID and the original reach ID are displayed.

Click **OK** or **Cancel** to exit this screen.
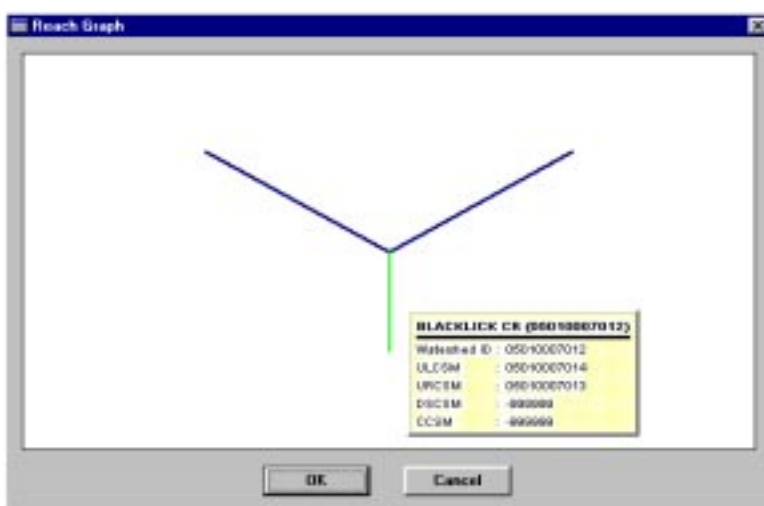
Screen 10.4.5

Tip: To add, remove, or move a reach segment, highlight the segment and click on the right mouse button. A list of three options is available—"Add Reach", "Remove Reach", and "Move Reach". Selecting "Add Reach" will open a window that prompts you to right-click on the reach to which the new reach will be connected. Do so and choose the desired connectivity relationship from the following menu of three options: "Position Reach Upstream Left of ...", "Position Reach Upstream Right of ...", or "Cancel". Selecting "Cancel" will stop the reach addition process. Selecting either of the first two options will open the "New Reach" window (Screen 10.4.6). Fill out the required information in this window. Click OK to save changes or Cancel to stop the addition process.

Selecting "Remove Reach" will open a window prompting you to delete the segment. To delete, select Yes; to not delete, select No.

Selecting "Move Reach" provides options similar those in "Add Reach".

For watersheds delineated using Reach File Version 3 data, the connectivity will frequently not be complete. You will need to manually connect the stream reaches using the visualization screen functions.
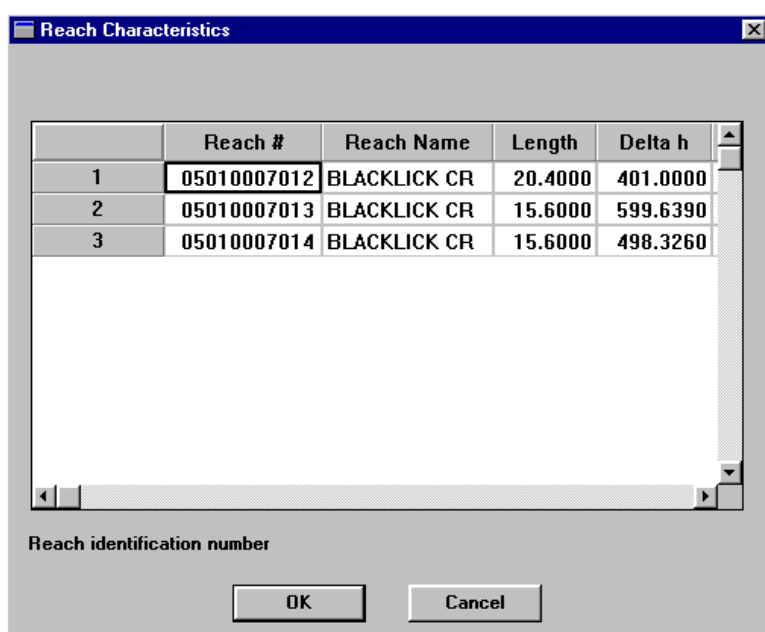
Screen 10.4.6

5. Select the **Reach Characteristics** button to view and edit reach parameters related to length and elevation. Data in this window (Screen 10.4.7) include Reach #, Reach Name, Length, Delta h, and Elevation. Length refers to the length of the reach segment (in miles). Delta h refers to the change in vertical elevation over the length of the reach (in feet). Elevation refers to the average elevation of the reach segment (in feet).

Tip: *Clicking on the column heading displays a definition of the parameter, as well as units. The same functionality exists in tables throughout the **NPSM** interface.*

Values in this table have been transferred directly from Reach File, V1 in BASINS. Values of -9999.000 are not acceptable and result from missing data in the database. These values must be edited directly on this screen before running the model.

Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.4.7

> *Tip:* Missing data can easily be generated using basic topographic maps. Missing data on this screen
> can be also be derived from other data sources in the BASINS GUI. Toggle between the NPSM
> interface and the BASINS GUI (if you have executed NPSM from within the BASINS View) by
> pressing the ALT and TAB buttons on your keyboard at the same time and selecting the ArcView
> icon. In BASINS View, turn on the DEM theme for your area of interest and the Reach File, V1
> theme. First be sure that the length values in the NPSM interface are representative of the
> reaches you are modeling. The length value extracted from BASINS is the length of the most
> downstream reach segment for each of the watersheds you previously delineated. If you did not
> develop your watershed's pour point to coincide with a reach segment node, or if you delineated a
> fairly large watershed with multiple Reach File, V1 stream reaches, you may need to measure a
> new segment length and put this value in the Length column of the "Reach Characteristics"
> window. The Measure tool in the BASINS GUI can be used to measure a new length. Missing
> elevation and delta h values can also be determined by identifying the elevation (using the DEM
> theme) at the most upstream end of your reach segment and at the most downstream end.
> Elevation is simply the average of the upstream and downstream elevation values, while delta h is
> the difference between these two values. These values should also be entered into the appropriate
> columns of the "Reach Characteristics" window.

6. Select the **F-Tables** button to display the function tables or rating curves used for flow calculations.
   If all required information for construction of these tables was available from the Reach File, V1
   database in BASINS, five rows of values for Depth, Area, Volume, and Outflow will be populated
   for each reach in your reach network (Screen 10.4.8). These tables must be complete for each reach
   in the network to run a successful simulation.

Display FTABLE for BLACKLICK CR (05010007012)

	Depth, ft	Area, acres	Volume, acre-ft	Outflow, cu. ft
1	0	106.43	0	0
2	0.10203	106.93	10.884	1.7419
3	1.0203	111.47	111.16	80.551
4	1.5304	121.56	168.67	158.1
5	1.913	340.73	298.31	199.28
6	2.2957	344.51	429.4	362.73
7	115.93	1468.5	1.0344E+005	1.2221E+006
8	229.57	2592.4	3.3417E+005	5.868E+006

Import/Export · Adjust Table Size · Cross Section · F-Curve · ☐ User-defined

Depth, ft

OK · Cancel

Screen 10.4.8

The relationship between the stream depth and flow rate can also be seen using the **F- Curve** button.
This curve can be displayed on various axis types (linear or logarithmic). Depth and flow rates can
be displayed for any point on the curve using the mouse (Screen 10.4.9). Depth and flow are
displayed on the left side of the screen. The units of the displayed values are the same as those on the
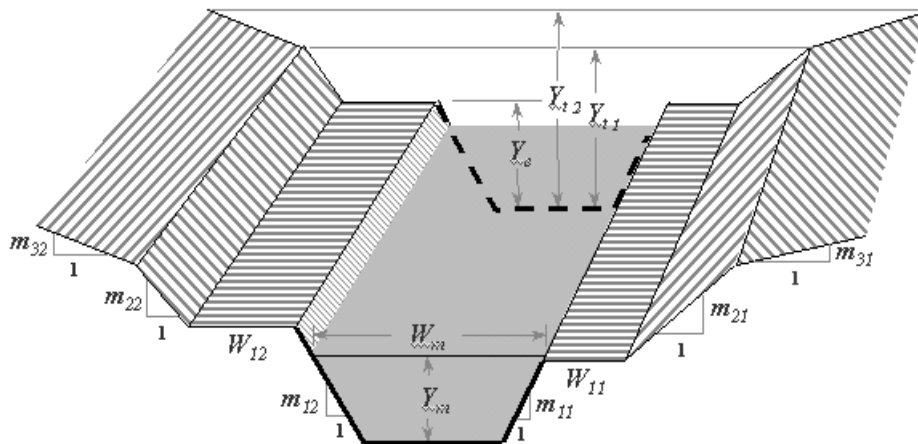axis scales.
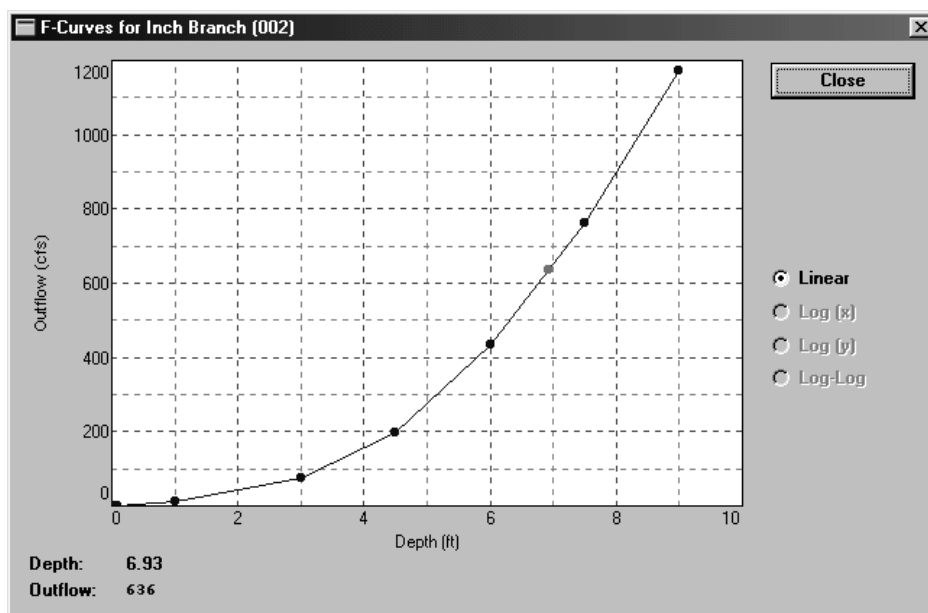
Screen 10.4.9

The stream cross section can be displayed using the **Cross Section** button. To adjust the F-Curve and F-Table to actual stream morphology, modify and update the cross section characteristics (Screen 10.4.10).
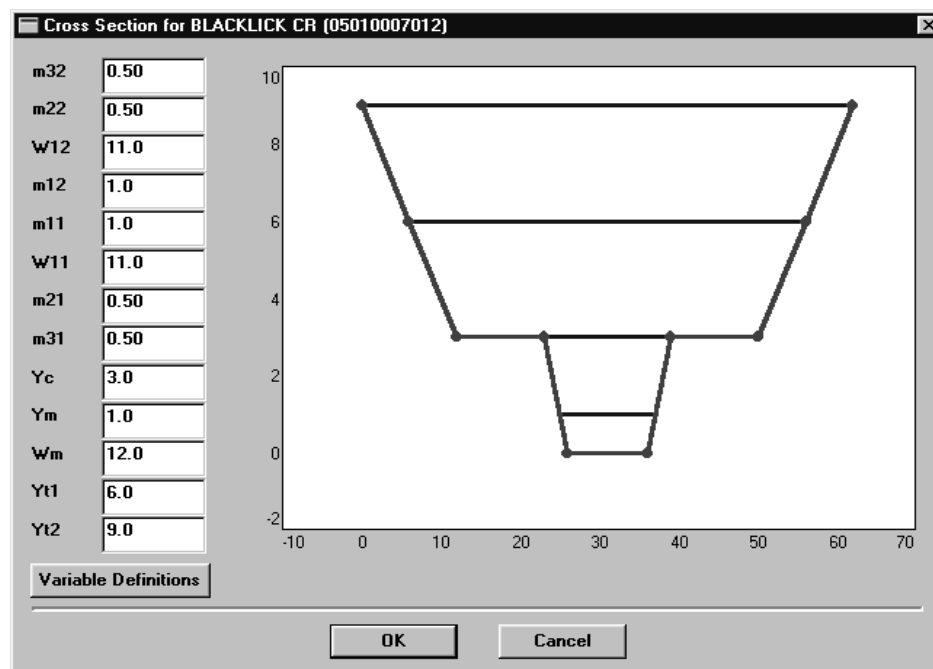
Screen 10.4.10

Figure 10.4.1

Use the left mouse button to vertically or laterally adjust various elements of the cross section. The vertical parameter table can also be used to adjust the cross section configuration. For assistance on cross section parameters, use the **Variable Definition** button (Figure 10.4.1).

The tables for each reach can be displayed by clicking the down arrow next to the "Display FTABLE for" box. These tables can be edited if a better stream representation is available. The number of rows can be adjusted by clicking on the "Adjust Table Size" button and making changes to the "Number of rows" box on the following screen (Screen 10.4.11). "Number of outflows" refers to multiple exits from your reach. Currently, NPSM supports a single outflow. Click **OK** to save changes and exit the "Adjust FTABLE Size" window or click **Cancel** to exit without changes.

If the F-Tables are not populated for each of the stream reaches, it is necessary to either develop them from scratch by using the "Adjust Table Size" option and providing the values directly in the F-Tables, or by using the Import option, which develops the F-Tables based on a limited set of stream characteristics.

Click on the **Import/Export** button to import the required data. The "F-Table Import/Export" window appears (Screen 10.4.12). Currently, only one option is available on this screen. This option is used to import stream characteristic data (from the Reach File Version 1 database in BASINS) to
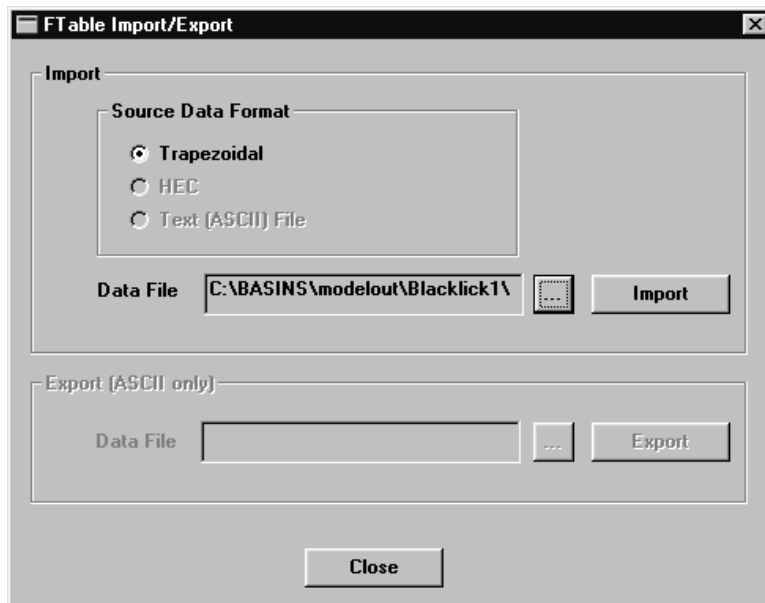

Screen 10.4.11

calculate F-Tables, while assuming a trapezoidal cross-sectional representation. Click the "**...**" button.

The subsequent window (Screen 10.4.13) prompts you to select the file to import. Be sure that the directory in the "Look in:" box matches the project name you defined for your project. This directory is located within the BASINS\MODELOUT\ directory. A file denoted by your project name and a .ptf extension should appear in the file list. Highlight this file and click **Open**.

After the file name appears in the "Data File" box, click the **Import** button.

> *Tip:* Check that the last row of the F-Table has a very large depth. This will ensure that the model does not calculate a depth out of range of the F-Table during the simulation. If this depth is too small compared to the depth at potential high flows, the simulation will fail. To change this depth by recreating the F-Table, use the Import method discussed below.

Screen 10.4.12

Select file to import

Look in: ☐ tutorial

🗋 tutorial.ptf

File name: tutorial.ptf

Files of type: Trapezoidal Files (*.PTF)

Open

Cancel
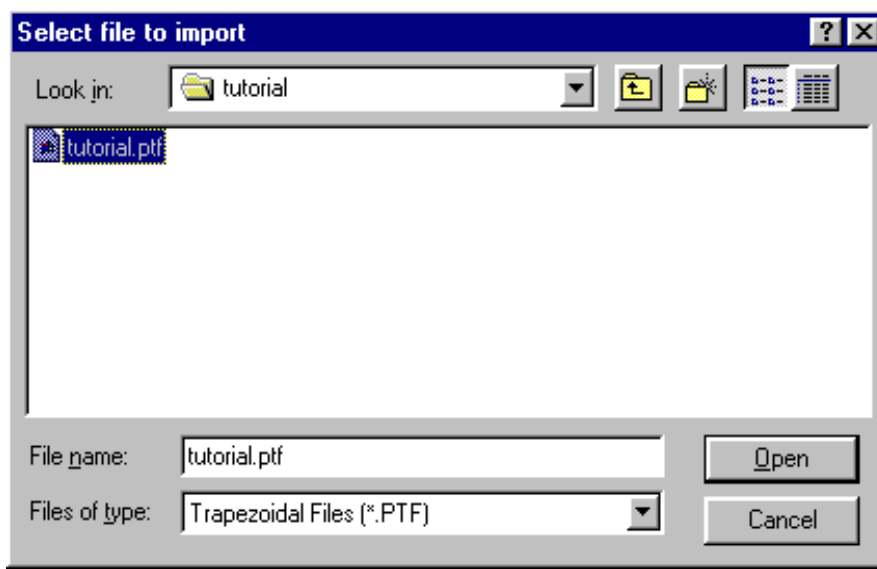
Screen 10.4.13

The "Trapezoidal Import Data Review" window appears (Screen 10.4.14). This window displays a table containing the following values for each reach: Reach ID, L (reach length, ft), Ym (mean reach depth, ft), Wm (mean reach width, ft), n (Manning's roughness coefficient, dimensionless), S (longitudinal reach slope, ft/ft), Type (trapezoidal, the only representation currently available), m32 (side slope of upper flood plain left), m22 (side slope of lower flood plain left), w12 (zero-slope flood plain width left, ft), m12 (side slope of the channel left), m31 (side slope of upper flood plain right), m21 (side slope of lower flood plain right) w11 (zero-slope flood plain width right, ft), m11 (side slope of the channel right), Yc (channel depth, ft), Yt1 (flood plain side slope changes at depth, ft), Yt2 (maximum depth, ft), Exits (# of exits), Fraction 1 (fraction of flow through Exit 1), Fraction 2 (fraction of flow through Exit 2), Fraction 3 (fraction of flow through Exit 3), Fraction 4 (fraction of flow through Exit 4), and Fraction 5 (fraction of flow through Exit 5). Currently, the Exits and Franction 1 values should be set to 1, because only one outflow can be simulated. Parameters defining the channel cross section are shown on Figure 10.4.1. Values for all of these variables must be appropriate to develop a working F-Table.

Click **OK** to save changes and proceed or **Cancel** to proceed without saving changes.

Screen 10.4.14

A message box will notify you that import is complete (if successful) or that F-Tables were not calculated for specified reaches (if unsuccessful). In either situation, click **OK** to continue. If F-Table calculation was unsuccessful, review the available data and repeat the import process.

Once the F-Tables for all reaches are complete, click **OK** to save changes and continue. Click **Cancel** to continue without saving changes.

7. Click **Done** to exit the "Reach Editor" window.

Tip: Clicking the cursor and dragging the line between column headings from left to right allows you to increase the width of any column in the table.

TUTORIAL

- *Select F-Tables.*
- *Select the Import/Export… option.*
- *Select "…" open the run1.ptf file located in BASINS\MODELOUT\TUTORIAL, and click Import.*

10.5 Simulation Time and Meteorological Data

1. Click this button to open the "Simulation Time and Meteorological Data" window (Screen 10.5.1). This window allows you to select the most appropriate meteorological data set and to define the model simulation period.

Screen 10.5.1

Meteorological data sets are compiled by weather station, in Watershed Data Management (WDM) files, which are binary files that contain the hourly data required by NPSM. In BASINS, a single WDM file containing meteorological data is provided for each state or territory. Each of these files contains meteorological data for up to 10 weather stations falling within that state or territory. If you wish to use meteorological data from a state outside your U.S. EPA Region, you can download WDM files from the BASINS download page or extract them from the Region CD of interest. The BASINS nation- and territory-wide WDM coverage includes data for 477 weather stations. Meteorological data for each weather station consist of hourly time series for air temperature, precipitation, dew point temperature, wind velocity, solar radiation, cloud cover, potential evapotranspiration, and potential surface evaporation. The period of record is generally January 1, 1970 to December 31, 1995, however, a number of stations contain shorter periods due to limited data availability.

At this point, be sure you have either downloaded or extracted a WDM file for your study area. The WDM data set consists of three files with the same name but different extensions (.WDM, .INF, .TXT). Each file name is a two-letter state abbreviation. The files should be located in the

project data directory in the MET_DATA subdirectory. If you do not have a WDM file, download the appropriate files from the web site and copy them to the MET_DATA subdirectory.

2. To select WDM files for application to NPSM, click the **Add...** button. The subsequent screen prompts you to add a WDM file to the project. Highlight the appropriate WDM file and click **Open**. Notice that the WDM file will appear in the "Select WDM file" box. Additional WDM files can be loaded in the same manner.

   All available weather stations for the WDM file in the "Select WDM file" box are listed in the "Weather station" box. Clicking the down arrow displays the entire list of stations. Weather stations are listed by the state abbreviation and the appropriate National Weather Service station name. Note that the time span of available data for the selected station is also displayed in this window.

3. Each watershed being simulated (those listed in the "Unassigned watersheds" box) must be assigned meteorological data from a single weather station. Separate watersheds may be assigned different weather station(s). You may assign a station to a given subwatershed based on how well the station represents the subwatershed's climate, taking into account proximity of the station to the subwatershed, elevation differences between the station and the subwatershed, and data availability. To assign a watershed, select the appropriate station in the "Weather station" box and double-click on the watershed name in the "Unassigned watersheds" box. The watershed and the corresponding weather station to which it has been assigned will appear in the "Assigned watersheds" box. To change a weather station designation already made, simply double-click on the watershed in the "Assigned watersheds" box, and it will be sent back to the "Unassigned watersheds" box.

Tip: *Toggle between NPSM and the BASINS GUI (if open) by pressing the ALT and TAB buttons on your keyboard simultaneously and selecting the ArcView icon. In the BASINS View, display WDM weather stations and examine their proximity to the selected watershed(s). Use the Identify tool to identify and determine the appropriate weather stations for modeling.*

   To write the meteorological time series data from a simulation (as model output), check the box next to "Write this station to PLTGEN file". This box can be checked for each weather station you use in your simulation. Upon running the model a separate file will be created for each station, and the file(s) will be located in the BASINS\MODELOUT\<project name> directory.

4. After selecting the appropriate weather stations, define a simulation period in the "Simulation time" box. The simulation period must lie within the time spans of all weather stations selected for watersheds. The format for specifying the start and end time is MM/DD/YYYY HH.

5. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

TUTORIAL

- *Open the tutorial.wdm file.*

- *Select the "PA Johnstown 2" weather station.*

- *Select to "Write this station to PLTGEN file".*

- *Assign the selected weather station to all three subwatersheds.*

- *Enter a simulation time period of 01/01/1980 (00) to 12/31/1984 (24).*

10.6 Land Use Editor

1. Click this button to view and/or modify the land use representation of watershed(s) being modeled. The "Land Use Editor" window (Screen 10.6.1) displays each pervious and impervious land unit defined for modeling. Recall that land uses in each watershed being modeled are divided into pervious and impervious land units based on the assigned perviousness percentages. The pervious land units are listed first for each watershed, followed by impervious land units. Each land unit is defined by Land Name (based on the classification of the land use used for modeling), Land Type (pervious or impervious), Area (land unit area, acres), and Watershed (the subwatershed to which that land unit is assigned).

   The land units in this table can be edited. Additional land units can be added and existing land units can be removed by clicking the right mouse button and performing the appropriate action. Additionally, Land Names can be changed by clicking on the appropriate cell and typing in a new name. Land Types can be changed from pervious to impervious or impervious to pervious by clicking on the appropriate cell and selecting the Pervious or Impervious radio button in the "Land Type" box at the top of the window. Land unit areas can be changed by clicking on the appropriate cell and typing in a new numerical value. Note that the "Current" watershed area in the "Total Watershed Area" box is updated as changes to land unit areas are made. The Watershed number for each land unit can also be changed by clicking on the appropriate cell and typing in a new number. This new Watershed number must match a watershed defined in the "Add/Remove Reaches" section of the "Reach Editor".

2. Click **Cancel** to exit this screen if you made no changes. Click **OK** to exit this screen if you made changes and would like to save them.
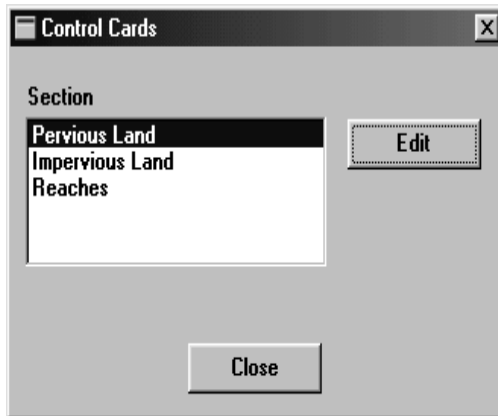

Screen 10.6.1

TUTORIAL

- *Observe the land units for each of the three subwatersheds.*

10.7 NPSM Control Cards

1. Click this button to select HSPF modules to run during this NPSM simulation. The "NPSM Control Cards" window (Screen 10.7.1) displays three options for editing—Pervious Land, Impervious Land, and Reaches. HSPF modules for each of these sections must be defined before running NPSM. Refer to *Hydrological Simulation Program - FORTRAN, User's Manual for Release 11.0* (Bicknell et al, 1996) for a discussion of HSPF modules.

Screen 10.7.1

2. Highlight "Pervious Land" and click **Edit**. The subsequent window, "Pervious Land Activity" (Screen 10.7.2), displays the HSPF modules for pervious land units available for simulation. Click an "X" in each box you wish to simulate. Modules in the Impervious Land and Reaches sections that correspond to those selected in this section must also be selected. For example, if Water Flow (PWATER) is selected for Pervious Land, IWATER must be selected for Impervious Land and HYDR and ADCALC must be selected for Reaches. Refer to the HSPF user's manual for a discussion of HSPF modules. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

3. Highlight "Impervious Land" and click **Edit**. The subsequent window, "Impervious Land Activity" (Screen 10.7.3), displays the HSPF modules for impervious land units available for simulation. Click an "X" in each box you wish to simulate. Modules in the Pervious Land and Reaches sections that correspond to those selected in this section must also be selected. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.7.2

 Screen 10.7.3

4. Highlight "Reaches" and click **Edit**. The subsequent window, "Reach Activity" (Screen 10.7.4), displays the HSPF modules for impervious land units available for simulation. Click an "X" in each box you wish to simulate. Modules in the Pervious Land and Reaches sections that correspond to those selected in this section must also be selected. Note that in some situations, selection of one HSPF module results in automatic selection and graying-out of another HSPF module. This occurs because some HSPF modules are required to run other modules. As an example, select GQUAL and notice that ADCALC is automatically selected and grayed-out. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.
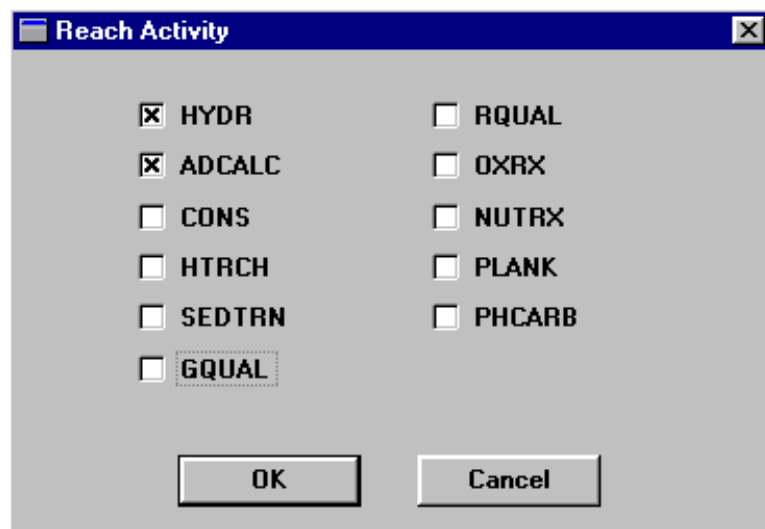
Screen 10.7.4

5. Click **Close** to close the "Land Use Editor".

TUTORIAL

- *Edit the Pervious Land. Select ATMP, PWATER, and PQUAL.*

- *Edit the Impervious Land. Select ATMP, IWATER, and IQUAL.*

- *Edit the Reaches. Select HYDR, ADCALC, and GQUAL.*

10.8 Pollutant Selection Screen

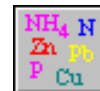1. Click this button to open the "Pollutant Selection Screen" (Screen 10.8.1) and select the pollutants for modeling. This window is divided into the following sections: Eutrophication Parameters; Gasses; General Quality; Pesticides 1, 2, 3; Tracer; and Sediment/Solids.

Screen 10.8.1

2. In the "Eutrophication Parameters" section, the Nitrogen Cycle and/or the Phosphorus Cycle can be selected for simulation. If you select either of these, the appropriate HSPF modules for Pervious Land, Impervious Land, and Reaches must be selected and the appropriate parameter values must be developed in the default data set. Refer to Section 10.15 for how to create and edit default data files.

3. In the "Gasses" section, Dissolved Oxygen and/or Dissolved Carbon Dioxide can be selected. If you select either of these, the appropriate HSPF modules for Pervious Land, Impervious Land, and Reaches must be selected and the appropriate parameter values must be present in your default data set.

4. In the "General Quality" section, up to three parameters from the Pollutant List can be selected for a single simulation. A pollutant is selected by highlighting its name in the "Pollutant list" and clicking on the right arrow button or by simply double-clicking on the pollutant name. Once selected, the pollutant will appear in the "Selected Pollutants" box. If you wish to unselect a selected pollutant, highlight the pollutant name in the "Selected Pollutants" box and click on the left arrow. If any General Quality pollutants are selected, the appropriate HSPF modules must be selected for Pervious Land, Impervious Land, and Reaches. Additionally, required parameters must be defined for the pollutant in a default data set.

Tip: Clicking the mouse on any name in the Pollutant List and typing any letter of the alphabet will move you to the list of pollutants beginning with that letter.

5. Up to three pesticides and one tracer can be simulated by clicking the box next to the name and entering a user-specified pesticide or tracer name. Once again, the appropriate HSPF modules must be selected for Pervious Land, Impervious Land, and Reaches and required parameters must be defined for the pesticide(s) and/or tracer in a default data set.

6. Sediment and solids can be simulated by checking the "Sediment/Solids" box. If this selection is made, you must click the **Distribution** button and populate the "Sediment/Solids Distribution" table (Screen 10.8.2).

   This screen requires you to enter Sand, Silt, and Clay fractions for sediment in each land unit in each watershed being modeled. This is done by clicking on a cell and entering a value between 0 and 1. The Sand, Silt, and Clay fractions must add up to 1 for each land unit. The appropriate HSPF modules must be selected for Pervious Land, Impervious Land, and Reaches and required parameters must be defined for sediment modules in a default data set. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.8.2

7. Once all pollutants have been selected for the simulation, click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Tip: To model a pollutant, the appropriate HSPF modules for Pervious Land, Impervious Land, or Reaches must be selected and an NPSM default file containing required pollutant parameter values must be developed. A starter default file packaged with BASINS contains uncalibrated parameter values for a number of General Quality parameters. The parameters for these pollutants have been defined for the BASINS default land use classification (the lumped Anderson Level II classification). Note that to simulate additional pollutants, it is necessary to set up and populate a default data set for appropriate pollutants, land use types, and reaches. Procedures for developing a default data set are defined in Section 10.15.

TUTORIAL

- *Select FECAL COLIFORM as the General Quality pollutant to model.*

10.9 Point Sources

1. Click this button to incorporate point source facility discharges into your simulation. The subsequent window (Screen 10.9.1) provides four options—**Setup, Pollutant Mapping, Loads,** and **Multipliers.**
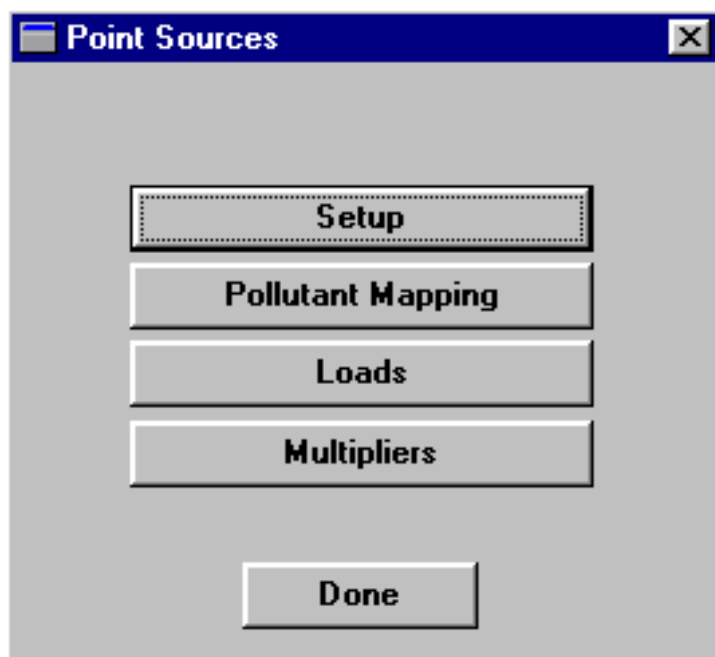
Screen 10.9.1

2. Click *Setup* to open the "Point Sources Setup" window (Screen 10.9.2). Observe which facilities discharge into the reaches being modeled. This is done by selecting each reach, one at a time, in the "Reach name" box and viewing the corresponding point source facility table. Any point source facility found in the Permit Compliance System database will be listed. The table includes Discharger Name, NPDES No., and Mile Point.

   In the event that this point source facility information needs to be edited, e.g. a facility needs to be added, a facility needs to be deleted, or information concerning a facility is incorrect, changes can be made from this screen.

   To add a new discharger to any reach being modeled, click the **Add** button. In the "Add Discharger" window (Screen 10.9.3), enter the discharger's name in the "Name" box, the discharger's NPDES number in the "NPDES No." box, and the facility's milepoint in the "Mile Point" box (in miles-from the most downstream point of the reach segment it discharges to). All pollutants the facility discharges should also be added on this screen. Add a pollutant by typing a pollutant name in the "Pollutant" box and clicking the **Add** button. The pollutant will be transferred from the "Pollutant" box to the large box below. Add remaining pollutants in the same manner. Remove selected pollutants by clicking on their names in the large box and clicking the **Remove** button. Click **OK** to save any changes and exit this window or **Cancel** to exit the window without saving changes.

Tip: Flow must be added as a pollutant parameter in the "Add Discharger" window.

To remove a discharger from the "Point Sources Setup" window, simply click on the discharger's name and click the **Remove** button.

To edit a discharger's name, NPDES number, mile point, or the names of pollutants it discharges, click on the discharger's name and click the **Edit** button. The "Edit Discharger" window opens and enables you to edit the current information. This window is identical to the "Add Discharger" window.

Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.9.2

Screen 10.9.3

3. Click **Pollutant Mapping** to open the "Point Source Pollutant Mapping" window (Screen 10.9.4). Due to the variation in pollutant naming conventions from different sources, it is necessary to ensure that all pollutants are properly represented in the model. The "Point source pollutants" box contains all pollutants monitored for facilities in the selected watershed(s). The "Modeled pollutants" box contains all pollutants selected in the NPSM Pollutant Selection window for this simulation. To model pollutant data from a point source facility, it is necessary to match the pollutant in the "Point source pollutants" box (by highlighting it) with the corresponding pollutant in the "Modeled pollutants" box (by highlighting it) and to click on the **Map** button. The matched pollutants will move to the "Mapped Pollutants" box. If you wish to change a designation, simply highlight the pollutant combination in the "Mapped Pollutants" box and click on the **Unmap** button. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.
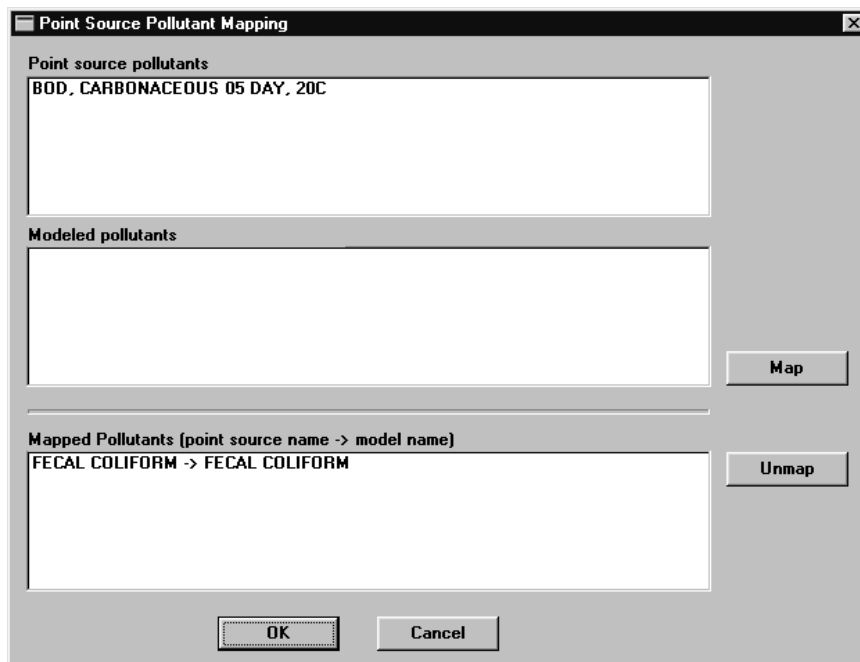
Screen 10.9.4

4. Click **Loads** to open the "Point Sources Pollutant Information" window (Screen 10.9.5). Discharger names for all point source facilities discharging into the stream reaches are listed in the "Discharger Name" box. Click the down arrow next to this box to view the entire list. Notice that the reach to which the facility discharges is identified below the discharger name. Point source contributions can be designated as either Constant or Time variable in the "Flow and Load" box. The "Active constituents" box displays flow and pollutants that have been mapped to the point source pollutant list for the facility listed in the "Discharger Name" box, but have not been selected for modeling. Select flow and pollutants you intend to model by highlighting their names in the "Active constituents" box and clicking the right arrow or simply by double-clicking on their names. To unselect a pollutant, highlight the appropriate name in the "Point source constituents" box and click on the left arrow.

Screen 10.9.5

To represent the point source contribution as constant, select the "Constant" radio button, be sure that flow and any pollutants you wish to model are located in the "Point source constituents" box, and click the **Edit Data** button.

The "Constant Flow and Load" window (Screen 10.9.6) appears. This window displays the Discharger Name and Reach for the facility selected on the previous screen. Additionally, it displays the flow and pollutant loading values for each pollutant selected for this facility. The flow and pollutant loading values can be edited directly on the screen as in other NPSM interface windows. Flow and pollutant loading units can be viewed by clicking on the respective column heading. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

Screen 10.9.6

To represent the point source contribution as time variable, select the "Time Variable" radio button, be sure that flow and any pollutants you wish to model are located in the "Point source constituents" box, and click the **Edit Data** button. The "Time Varying Flow and Load" window (Screen 10.9.7) appears. This window displays the Discharger Name and Reach for the facility selected on the previous screen and time-varying flow and load values in a tabular format. The first column contains the number of each time-variable record used to represent the discharge data. The Date and Time columns contain the date and time, respectively, for each time-variable record. The Date column uses a "MM/DD/YYYY" format for date entry, and the time column uses a "HH:MM" format for time entry. The remaining columns contain the flow and additional pollutant loading values for each time

variable record. Flow and pollutant loading units can be viewed by clicking on the respective column headings. Values for Date, Time, flow, and pollutant loading can be entered directly into the table. In lieu of manually entering all time series data, data can be imported into this table using the **Import** button. Clicking the **Import** button will enable you to select an HSPF MUTSIN file containing your time series flow and pollutant loading data. In order to use this option, you must develop a MUTSIN file. Refer to *Hydrological Simulation Program - FORTRAN, User's Manual for Release 11.0* for information regarding the MUTSIN file format. Click **OK** to save changes and exit this window or **Cancel** to exit this window without saving changes.

Screen 10.9.7

In the event that you do not wish to enter a time variable value for every time step in your simulation period, select an option from the "Options for handling incomplete time series data" drop-down list. Available options include: stop on missing data, fill missing data with 0.0, fill missing data with -1.0E30, and fill missing data with next value.

Click **OK** to save any changes and leave the "Point Source Pollutant Information" window or **Cancel** to leave the "Point Source Pollutant Information" window without saving changes.

5. Click **Multipliers** to open the "Flow and Load Multipliers" window (Screen 10.9.8). This window is normally used to evaluate different point source contribution scenarios or to predict the impact of future increase in point source discharges (existing sources or new discharge) of an existing discharge on the overall loading. For each point source facility (listed in the "Discharger Name" box), you can assign different flow and pollutant load multipliers. The Default multiplier value is 1.0. However, if you wish to increase or decrease the current flow, loading, or both by a certain

Screen 10.9.8

factor, simply enter this factor directly into the table. Click **OK** to save any changes and leave this window or **Cancel** to leave the window without saving changes.

6.   Click the **Done** button on the "Point Sources" window to continue.

TUTORIAL

- *Recall that there are no data in PCS for the facilities in the three subwatersheds. Therefore we will add a facility for demonstration purposes.*

- *On the "Point Sources Setup" screen, be sure the "Reach name" is set to Blacklick Cr. (0501000712), and click the Add button.*

- *On the "Add Discharges" window, enter Blacklick C. Point Source #1 as the discharges name, US1234567 as the NPDES number, 1.0 as the Mile Point, and flow and fecal coliform as the pollutants.*

- *On the "Point Source Pollutant Mapping" screen map fecal coliform.*

- *On the "Point Source Pollutant Information" screen select constant flow and load, be sure that both flow and fecal coliform are on the "Point Source constituents" box, and select Edit Data. Enter a flow of 0.2629 cfs and a fecal coliform loading of 504,500 #/hr.*

10.10 Default Data Assignment

1. Click this button to open the "Default Data Assignment" window (Screen 10.10.1). This window enables you to select an NPSM default data file and assign default data to land units, reaches, and pollutants in the current project. First, select a default file by depressing the button denoted by the ellipsis points (**...**). The subsequent window prompts you to select a file. Generally, NPSM default files are saved in the BASINS\MODELS\NPSM directory. They are denoted by a .DEF file extension. The starter default data file packaged with BASINS is named STARTER.DEF. Section 10.5 discusses how to create and modify your own default data files. Open the default file.
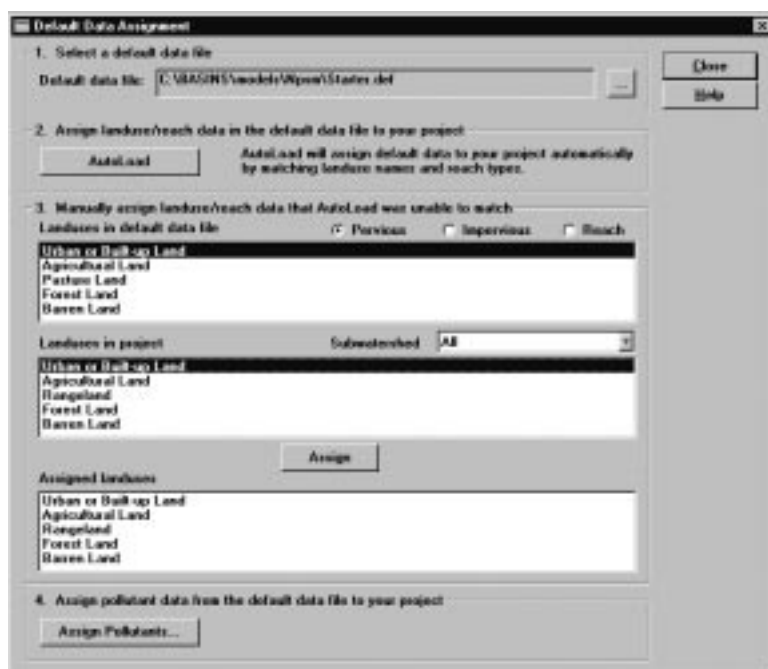
Screen 10.10.1

2. Land use, reach, and pollutant data from the default file must be assigned to each land unit, reach, and pollutant being modeled. Click the **AutoLoad** button to automatically assign default data from the default file you've selected to your current project.

    Clicking the **AutoLoad** button searches the default data file for each land unit name, reach name, and pollutant name being modeled.

    Default data are automatically assigned for each modeled watershed land unit name that has a land use name match in the default data file. If the default data file contains no data for one or more of the land units being modeled, a warning window appears (Screen 10.10.2) notifying you to which land unit(s) no data have been assigned. For example, if three watersheds being modeled each contain a Rangeland land unit and there are no data available for Rangeland land use in the default data set, a warning will appear notifying you that three separate Rangeland land units were not assigned default data. In this situation, another land use type may be manually assigned to each of these three Rangeland land units (as discussed below in section 10.10.4.3). Click **OK** to exit the warning window.
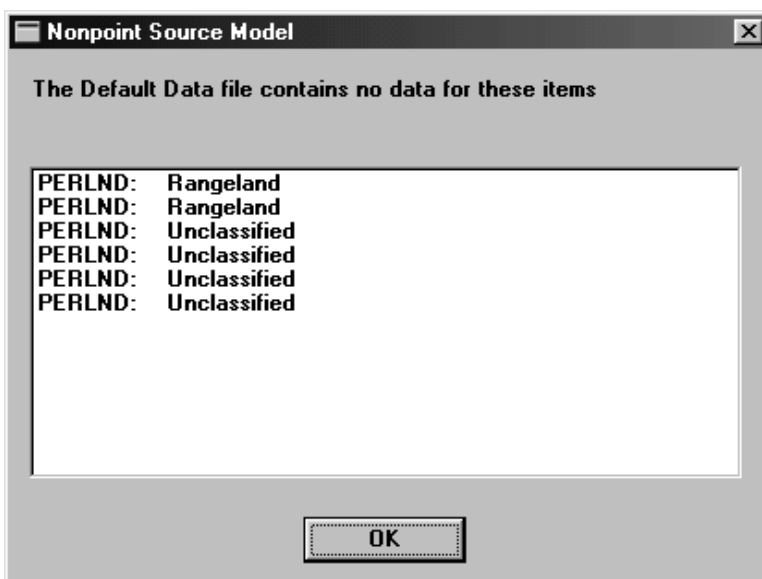
Screen 10.10.2

Each reach being modeled is automatically assigned data from the first reach listed in the "Available Data" box unless an identical name match is found in the default data set (in which case the latter is used). Note that when any land units or reaches have been assigned, their names appear in the "Assigned Land Units" box.

Pollutants selected for simulation are also searched for in the default data file. Where matches are found, default data are automatically assigned. Where no pollutant name matches are found, a warning is given (Screen 10.10.3). Click **OK** to exit this window. In these situations, the pollutant data must manually be assigned data from the default data set. This is also discussed below.
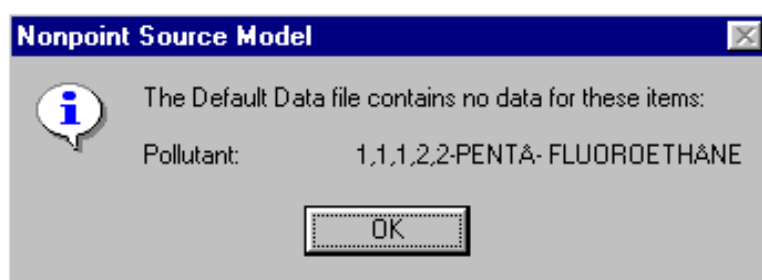
Screen 10.10.3

3. Once **AutoLoad** has assigned data for land unit, reach, and pollutant matches found in the default file, data can manually be assigned for the remaining land units, reaches, and pollutants.

   Select a radio button for Pervious [Land], Impervious [land], or Reach(es) to display land units or reach groups corresponding to that category. The "Landuses in default file" box contains land use categories for which default data are contained in the selected NPSM default data file. The "Landuses in Project" box contains land units that make up the watershed(s) being modeled. For reaches, the "Reaches in default data file" box contains the reaches/reach type for which default data are available. The "Reaches in Project" box contains the reach(es) being modeled in watershed(s) of interest.

To assign default data for any pervious land units without assigned data, first click the radio button next to "Pervious". Highlight a land unit name in the "Landuses in default data file" box and highlight the name of an unassigned land use name in the "Landuses in project" box. The land use name you select in the "Landuses in default datafile" should be most representative of the land use selected in the "Landuses in project" box. In the "Subwatershed" box, use the drop-down menu to select the watershed(s) you want to use this assignment. Click the **Assign** button. Notice that the land unit name you selected from the "Landuses in project" box appears in the "Assigned Landuses" box. Perform this step for each unassigned pervious land unit.

Next, click the radio button next to "Impervious Land". Follow the same procedure used to assign data for pervious land units. Perform this procedure for each unassigned impervious land unit.

Click the radio button next to "Reach". Highlight a reach name in the "Reaches in default data file" box, highlight any unassigned reaches in the "Reaches in project" box, and click the assign button. Notice that the reach name you selected from the "Reaches in project" box appears in the "Assigned reaches" box. Perform this step for each unassigned reach.

4. Click the "**Assign Pollutants ...**" button to manually assign data for any pollutant data not assigned automatically with the **AutoLoad** button. Doing so opens the "Default Pollutant Assignment" window (Screen 10.10.4). The "Default data file contains" box lists all pollutants defined and parameterized in the default data file by category (General Quality, Pesticide, Tracer). The "This project contains" box lists all pollutants that have been selected for modeling (in the NPSM Pollutant Selection window). The "Available conventional pollutant data" list indicates whether the selected default data set contains data for conventional pollutants. Conventional pollutants with data will appear in black letters while those without data will appear grey. Data for these parameters are needed when sediment (SEDMNT, SEDTRN), nutrient (NITR, PHOS, NUTRX), or dissolved oxygen (PWTGAS, IWTGAS, OXRX) HSPF modules are selected in the "NPSM Control Cards" (Refer to section 10.7). Default data, when available, are automatically assigned to these standard conventional pollutants. For any unassigned pollutants, highlight a pollutant name in the "This project contains" box, and highlight the corresponding/appropriate pollutant name in the "Default data file contains" box. Click on the **>>Assign>>** button, and notice that the pollutant name appears in the "Assigned Pollutants" box. Repeat this step for each unassigned pollutant. Click **Close** to exit this window.

5. After assigning default data to all pervious and impervious land units, reaches, and pollutants, click **Close** to save changes and exit the "Default Data Assignment" window.

Tip: *If you accidentally assign the wrong type of land use data to a land unit, simply repeat the assignment procedure using the correct land use data.*

Tip: *To observe what land use default data have been assigned to each land unit, double-click on each land unit name (one at a time) located in the "Assigned Land Unit" box.*
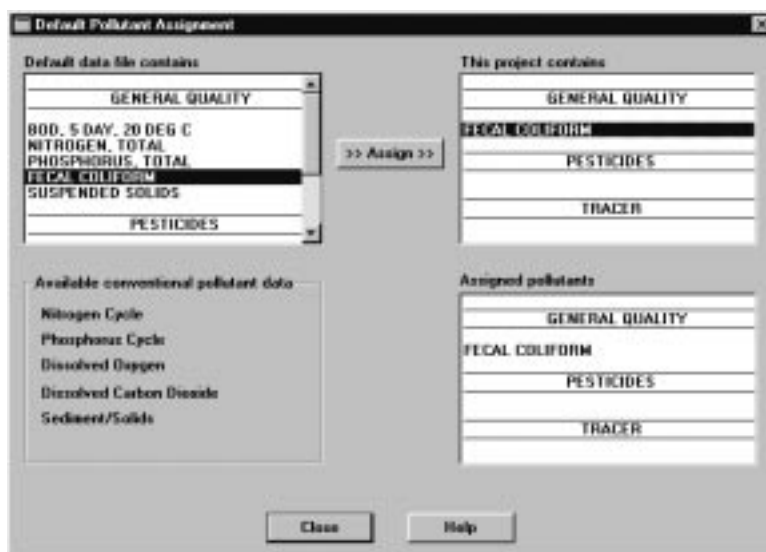
Screen 10.10.4

Tip: If you do not want to assign data from the default file using **AutoLoad** button, it is possible to manually assign each pervious land, impervious land, reach, and pollutant.

TUTORIAL

- Select the starter.def default file.
- Use **AutoLoad** to load the default data from the default file into the current project.

10.11 Input Data Editor

1. Click this button to begin populating all input parameter fields in the "NPSM Input Data Editor" (Screen 10.11.1). In this window, you can access and edit each model parameter and corresponding value including the default parameter values from the default data file. The procedure for accessing and editing the model parameters is described in this section.

   The "Input Data Editor" window displays two sections. The upper section contains a hierarchical list of the module sections and parameters of the HSPF model. In this screen (screen 10.11.1), the three major modeling segments used to subdivide the watershed are shown—pervious land units (PERLND), impervious land units (IMPLND), and stream reaches (RCHRES). The lower section is a status box that displays the definition of the highlighted module or parameter. PERLND (Pervious land) contains all simulation modules associated with pervious land, IMPLND (Impervious land) contains all the simulation modules associated with impervious land units, and RCHRES (stream reaches and reservoirs) contains all the in-stream simulation modules.
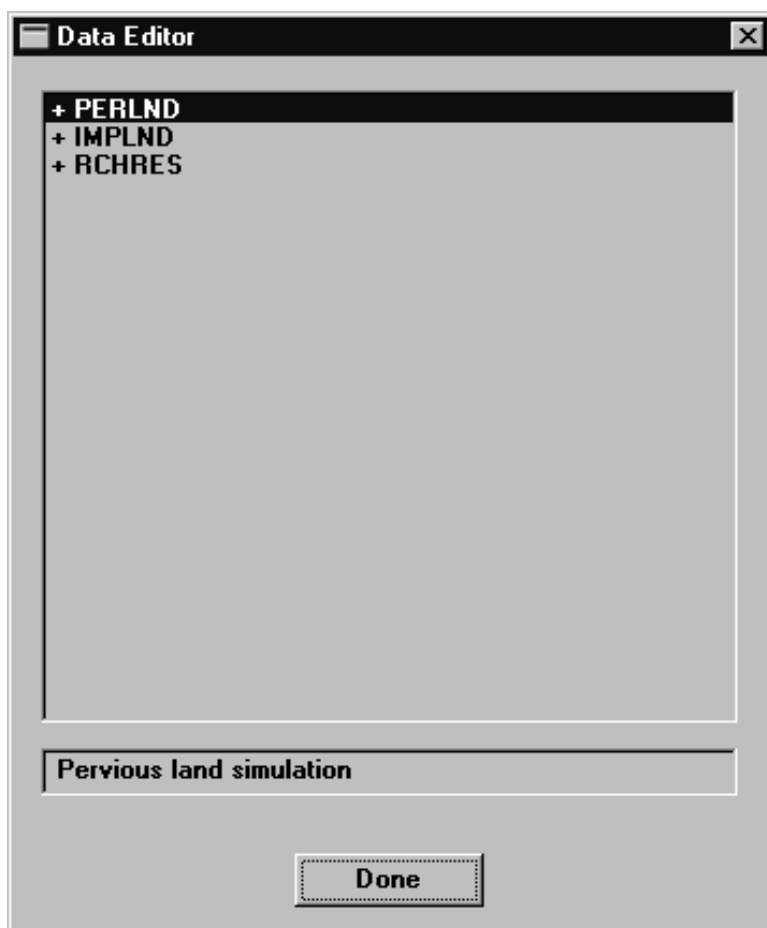
Screen 10.11.1

2. To view data for each of these model segments, double-click on a segment name. Notice that the "+" symbol next to the segment name becomes a "-" symbol and a list appears below the segment name.

A "+" sign indicates that the item is expandable and more options are hidden under it. A "-" sign indicates that the item is already fully expanded. Double-click on an expanded item to hide the options under it.

For example, when "+PERLND" is double-clicked, it becomes "-PERLND" and the following list is shown: ATMP, SNOW, PWATER, SEDMNT, PSTEMP, PWTGAS, PQUAL, MSTLAY, PEST, NITR, PHOS, and TRACER. This list contains all of the HSPF modules associated with the segment name (PERLND, in this case). Each HSPF module that was not previously selected in the "NPSM Control Cards" window is designated with an "[N/A]". If, for example, you did not select to simulate snow in the "NPSM Control Cards" window, it appears as "+[N/A] SNOW" in this window. Observe also that these module names are designated with "+" symbols, meaning options are hidden.

3. Make sure "PWATER" has been selected in the "NPSM Control Card" window (i.e. "PWATER is preceded by a "+" rather than a "+[N/A]"). Double-click on "+PWATER" to expand the module/parameter list. The list under "PWATER" contains PWAT-PARM1, +PWAT-PARM2, +PWAT-PARM3, +PWAT-PARM4, +Monthly Input Parameters, and +PWATER-STATE1 (Screen 10.11.2). These are the major HSPF groups associated with the PWATER module. Notice that the first HSPF group, "PWAT-PARM1", is not designated with a "+" sign. Any item without a "+" or "-" designation is a data item. Double-clicking on a data item opens its editor window.
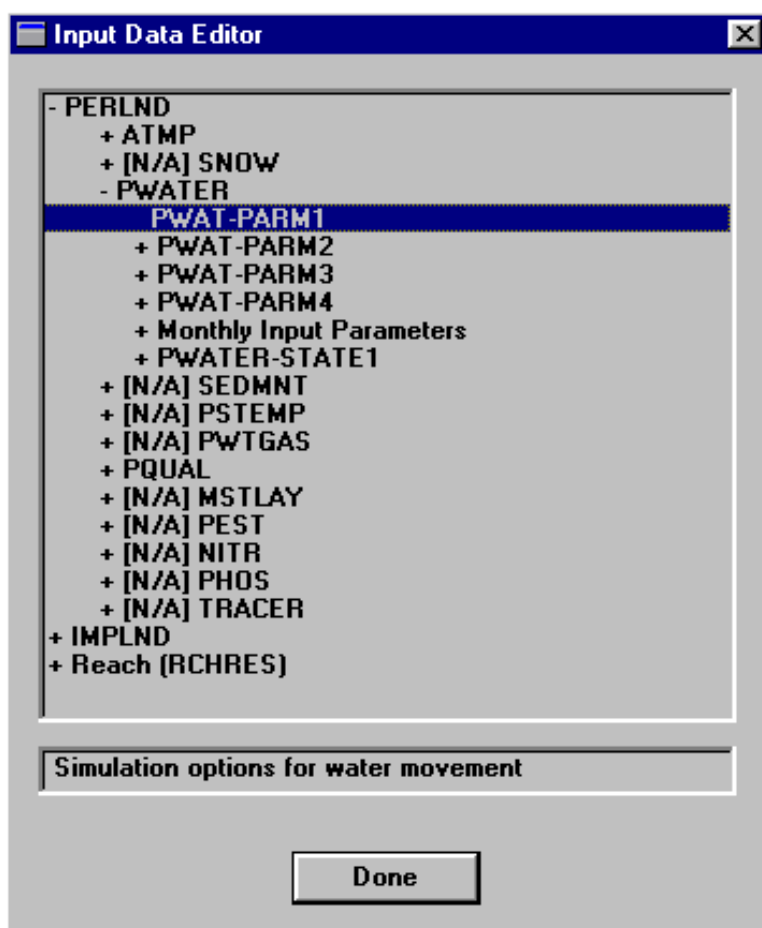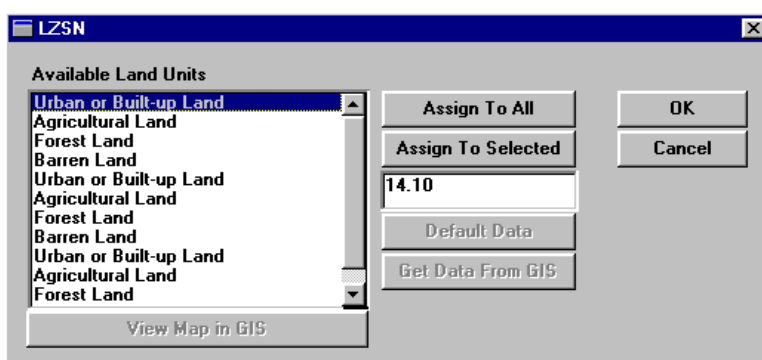

Screen 10.11.2

4. Double-clicking the "PWAT-PARM1" data item opens the "Simulation and Input Options (PWATER)" window (Screen 10.11.3). All current assignments on this window were made previously through default data assignment. Additional changes to any of these assignments can be made directly on the screen. Any changes made to these settings will be saved for the current project only. Click **OK** to save changes and exit the screen or **Cancel** to exit the screen without saving changes.

Tip: Changes made to one parameter screen may require changes in another screen. For example, if a parameter is represented in the model as constant over time and you decide to convert its time-varying option to monthly, it is also necessary to provide the monthly values. Refer to Hydrological Simulation Program - Fortran, User's Manual for Release 11.0 (Bicknell et al., 1996) for a more detailed explanation of the model structure, data needs, and review of the simulation options.

Screen 10.11.3

5. Note that the remaining HSPF group names under "PWAT-PARM1" are designated with "+" signs and therefore have more information hidden. Double-click on "+PWAT-PARM2". Notice that a list containing FOREST, LZSN, INFILT, LSUR, SLSUR, KVARY, and AGWRC appears. These are actual variable names that fall into the "PWAT-PARM2" HSPF group. They are data items and can be edited by double-clicking on the appropriate name.

6. Double-click on "LZSN" to open its editor window (Screen 10.11.4). Note that every land unit being simulated is listed in the "Available Land Units" box (only pervious land units, because this is the PERLND section). Highlighting each land unit name displays the assigned parameter value. These parameter values can be edited directly into this window for each land unit. Clicking **Assign to All** will assign the current value to all land units being simulated. Clicking **Assign to Selected** will assign the current value to the highlighted land unit. Click **OK** to save changes and exit the screen or **Cancel** to exit the screen without saving changes.

Screen 10.11.4

7. A similar procedure can be performed for every HSPF parameter in the PERLND, IMPLND, and RCHRES segments. To exit the "Input Data Editor", click **Done.**

Tip: Individual data editor windows vary, but the functionality remains consistent. For pollutant-related variables, keep in mind that "Assign to All" assigns the current value for the selected pollutant to every possible land unit or reach, as well as every possible pollutant. "Assign to All Constituents" assigns the current value to the selected pollutant and all other pollutants for the selected land unit or reach and all other land uses or reaches for the selected pollutant.

Tip: Appendix B of this manual contains an HSPF data dictionary that includes HSPF parameter names, definitions, units, and minimum and maximum acceptable values. This is a valuable resource for populating the "Input Data Editor". However, remember that the appropriate values for one land unit or a given pollutant might not be appropriate for another land unit or pollutant.

Tip: Any changes made within the "Input Data Editor" apply to only the current project. If you wish to make permanent changes, it is necessary to edit the default data file.

Tip: Beware of reassigning default data in the "Default Data Assignment" window after editing parameter values in the "Input Data Editor" window. Doing so will negate changes made in the "Input Data Editor". That is, changes made to land units, reaches, or pollutants in the "Default Data Assignment" screen will be saved over prior changes made in the "Input Data Editor" window.

TUTORIAL

- *All data required for the current project are contained within the starter.def file. No changes need to be made in the "Input Data Editor".*

10.12 Output Manager

1. Click this button to specify the simulation parameters to print, the print intervals, and the grouping of output parameters in the "Output Manager" window (Screen 10.12.1). Although this system option is designed to enable output of potentially each HSPF parameter at the watershed, sub-watershed, land unit type, or reach level for various print intervals, it is advisable to limit the selection of print options to only those options needed.

Screen 10.12.1

2. For demonstration purposes, assume that you want to output (into a single file) the daily mean surface outflow for agricultural pervious land in a single watershed. The first step is to select an HSPF modeling "Module" by clicking one of the radio buttons next to Pervious, Impervious, or Reaches. These modules refer to pervious land units, impervious land units, and reaches, respectively. For this example, select "Pervious".

3. The "Sum landuse contribution in" box contains all watersheds being modeled in the current simulation. Use this box to select the watershed for which land unit contributions are summed. For this example, choose a watershed.

4. The "Landuse" box contains all available land units for the selected module. Use this box to select the land unit level at which output will be printed. "All Pervious" combines contributions from each pervious land unit type in your simulation. For this example, select "Agricultural Land".

5. Note that an output file name is automatically assigned.

6. Enter a directory in which to save this output file in the "Output folder" box. Be sure to key in the entire path (i.e., C:\BASINS\MODELOUT\<project name>).

7. In the "Print Interval" box, select one of the print interval options for output. Recall that NPSM operates on an hourly time step; therefore, you can choose from Hourly, Daily, Monthly, and Yearly. For this example, select "Daily".

8. In the "Type" box, you can select either Mean-valued or Point-valued. Mean-valued provides a mean value for each interval of the selected print interval, while Point-valued provides the value on the last hour of each print interval. For this example, select "Mean-valued".

9. The final step is to select the variable or variables to print. Output for all selected variables will be contained within a single file. The "Select a computed variable to print" box works in the same manner as the "Input Data editor". A "+" indicates that the item is expandable and more options are hidden under it; a "-" indicates that the item is already fully expanded. A parameter for which output can be printed contains no "+" or "-" symbol (analogous to the data items in the "Input Data Editor"). To select an output parameter, highlight its name in the "Select a computed variable to print" box, and click the >> button. Notice that the selected parameter moves to the "Selected variables" box. To unselect a parameter, highlight its name in the "Selected variables" box and click the << button. You may select up to 10 parameters for a single file. If you select more than 10 parameters, the additional parameters will be written to a new file. This new file will be located in the same directory designated for the first file. Also note that the definition and units of each parameter are listed below the "Select a computed variable to print" box when the parameter is highlighted. Select SURO (Surface outflow) for this example.

10. Following these procedures for the example would output the daily mean surface outflow for agricultural pervious land in all of the watersheds being simulated.

11. The same procedure must be performed to output parameter data for the Impervious and Reach modules. The Impervious module selections are similar to those for Pervious land, whereas the Reach selections are somewhat different. When selecting the radio button for Reach, note that the "Sum landuse contribution in" box becomes grayed out. You are given only the option to select a reach/watershed from the "Subwatershed" box. The "Subwatershed" box contains all reaches being simulated in the current project.

12. Click **OK** to save changes and exit.

TUTORIAL

- *Select Pervious as the Module.*
- *Select 05010007012 (Blacklick Cr) in the "Sum landuse contribution in" box.*
- *Select Agricultural land as the Landuse.*
- *Note the Output file name is P#_Agric.000.*
- *Select a Daily print interval.*
- *Select Mean-valued as the Type.*
- *Select the output folder as BASINS\MODELOUT\TUTORIAL\10007012.*
- *In PERLND, PWATER, select PERO (Total outflow from PLS) by highlighting the variable and clicking the >> button.*
- *In PERLND, PQUAL, highlight and select SOQUAL (Total outflow of QUAL from PLS).*
- *Select Forest land as the Landuse.*
- *Note the Output file name is P#_Fores.000.*
- *In PERLND, PWATER, highlight and select PERO (Total outflow from PLS).*
- *In PERLND, PQUAL, highlight and select SOQUAL (Total outflow of QUAL from PLS).*
- *Select Reaches as the Module.*
- *Select Blacklick Cr. (05010007012) in the Subwatershed box.*
- *Note the Output file name is R#_Black.012.*
- *Select the output folder as BASINS\MODELOUT\TUTORIAL\REACHES.*
- *In RCHRES, HYDR, highlight and select RO (Total rate of outflow from RCHRES).*
- *In RCHRES, GQUAL, highlight and select DQAL (Dissolved concentration of QUAL).*
- *Click OK to save changes and exit. You will be prompted to create new directories. Click Yes.*

10.13 Run NPSM

1. Click this button to execute **NPSM**. The model run is performed by executing a DOS-based program, HSPF Version 11.0.

2. When the model run is complete, you will be asked whether you want to view the output. Click **Yes** to view the output or **No** to return to the NPSM interface. Regardless of your selection, you will be able to view the output at any time by clicking the "View Output" button.

Tip: Remember to save the project before running the model. Save the current project by clicking the **Save the current project** button. Also be sure to note the names and locations of project files and output files for the current simulation run.

TUTORIAL

• *Run NPSM.*

• *Click **Yes** to view the output.*

10.14 View Time Series Output

Click this button to display output from the model run in either a text or graphical format.

The "View Time Series Output" window (Screen 10.14.1) provides options to view data for Pervious land units, Impervious land units, Reaches, or Weather Station data. Click the radio button beside the name of the module for which you wish to view data. You will be able to view only data for the parameters and corresponding land units or reaches you chose in the "Output Manager" window prior to executing the model or, in the case of weather stations, only data for the stations you chose to print output for in the "Simulation Time and Meteorological Data" window.

For example, if you selected to view only daily mean surface outflow for agricultural pervious land in all of the watersheds being simulated, you would be able to view only "Agricultural Land" in the "Pervious" list.
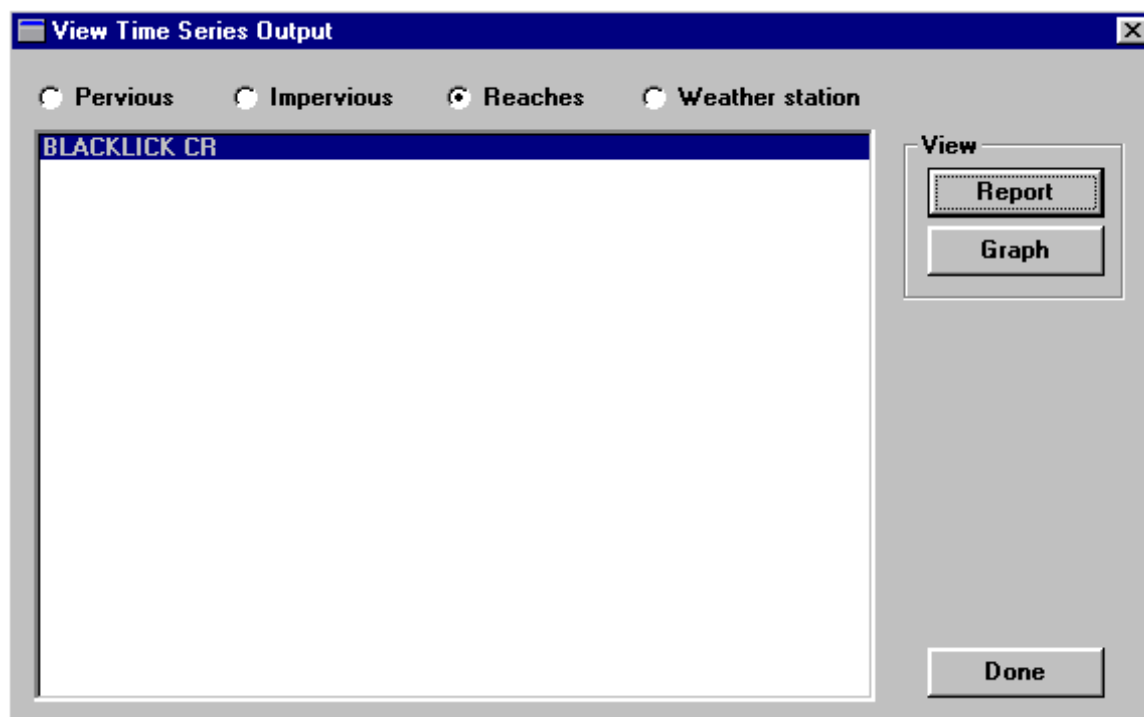
Screen 10.14.1

Viewing Output in a Text Format

1. View output in a text format by highlighting the appropriate Pervious or Impervious land unit, Reach, or Weather Station from the Pervious, Impervious, Reach, and Weather Station lists and clicking the **Report** button in the "View" box. The output file for the selected land unit, reach, or weather station is automatically displayed in the WordPad text editor. The file name is listed in the top left corner of the WordPad window. A sample output file is shown in Screen 10.14.2. The text file viewer can be changed by selecting *Select Output File Viewer...* from the *Project* menu heading in the NPSM interface.

```
R0_black.012 - WordPad
File  Edit  View  Insert  Format  Help

0501 HSPF FILE FOR DRIVING SEPARATE PLOT PROGRAM
0501 Time interval: 1440 mins          Last month in printout year:  9
0501 No. of curves plotted:  Point-valued:  0   Mean-valued:  2   Total  2
0501 Label flag:  0           Pivl:   24          Idelt:   60
0501 Plot title:    05010007012:  BLACKLICK C, Flow and Conc
0501 Y-axis label: cfs, mg/L, #/100ml
0501 Scale info:  Ymin:  0.00000              Threshold:-0.10000E+31
0501             Ymax:   1000.0
0501             Time:   20.000     intervals/inch
0501 Data for each curve (Point-valued first, then mean-valued):
0501 Label                    LINTYP    INTEQ    COLCOD      TRAN    TRANCOD
0501 RO (1)(1)                   0        5        1        AVER        2
0501 DQAL (FECAL COLI            0        5        1        AVER        2
0501
0501
0501
0501
0501
0501
0501
0501
0501 Time series (pt-valued, then mean-valued):
0501
0501 Date/time                    Values
0501
0501  1979 12 31 24  0  -0.10000E+31   -0.10000E+31
0501  1980  1  1 24  0    5.5105        10.572
0501  1980  1  2 24  0   14.074          6.2327
```

Screen 10.14.2

A brief description of important information in the output file (by line number) follows:

- Line 2 lists the selected print time interval and the end month of the simulation period.

- Line 3 lists the number of point- and mean-valued and total time series in the output file.

- Line 5 lists the plot title.

- Lines 12 to 21 list the parameters for which time series data are displayed in the output file. The order of these parameters corresponds to the order of the columnar data beginning at Line 27 and Column 6. These columnar data, beginning at Line 27 and going to the end of the file, are the time series output for the selected period, printed at the selected time interval. Column 1 is the year, Column 2 is the month, and Column 3 is the day. Column 6 is the data for the parameter listed in Line 12. Columns 7 to 15 (if output) contain the data for the parameters listed in Lines 13 to 21, respectively.

2. To stop viewing the output file, either click the **X** in the top right corner of the window or select *Exit* from the *File* menu heading. This returns you to the "View Time Series Output" window.
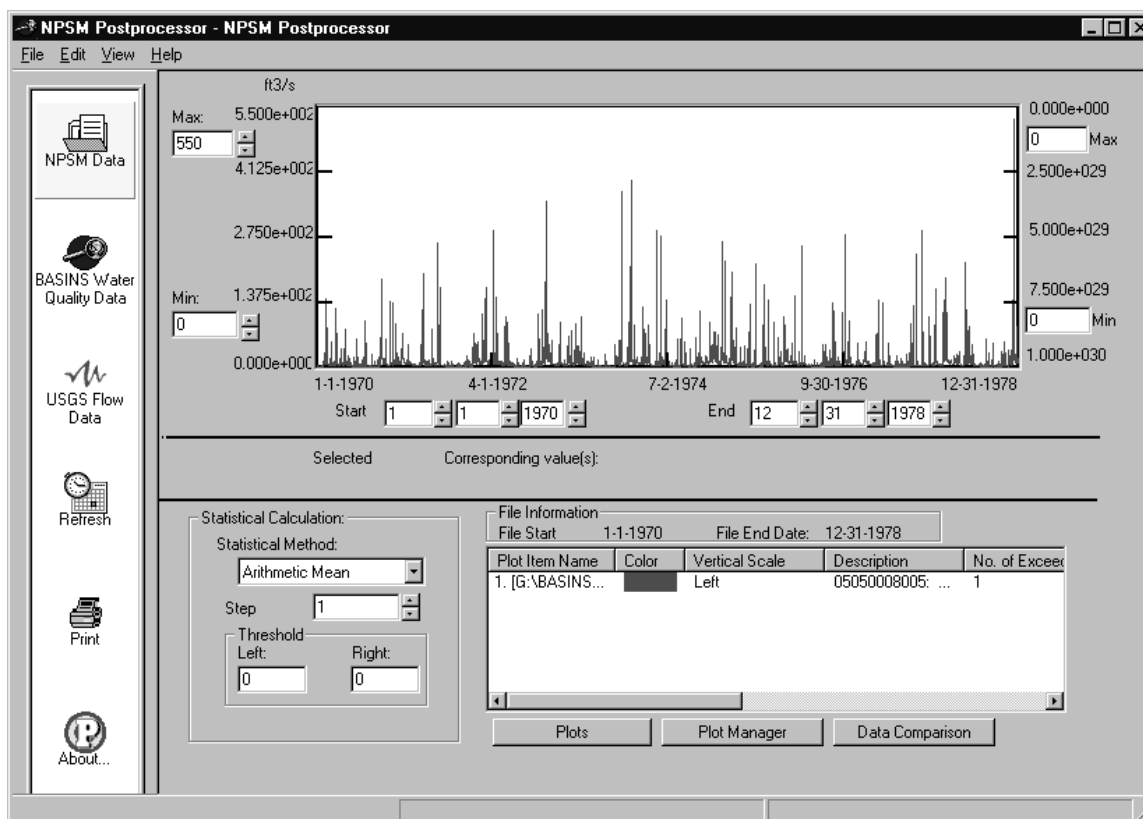

TUTORIAL

- *Click the Reaches radio button.*

- *Highlight Blacklick Cr.*

- *Select **Report** to view the text version of the model output for Backlick Cr.*

- *Exit the text window by clicking the **X** in the top right corner.*

Viewing Output Graphically in the NPSM Postprocessor

You can view output in a graphical format for a Pervious land unit, an Impervious land unit, or a Reach by highlighting the appropriate land unit or reach and clicking the **Graph** button from the "View" box (Screen 10.14.1). This executes the NPSM postprocessor. The postprocessor supports daily, monthly, and annual NPSM output. Weather Station data from NPSM is in an hourly format and therefore cannot be viewed in the postprocessor.

The NPSM postprocessor displays NPSM simulation output, BASINS water quality observation data, and USGS flow data in a graphical format. It also performs basic statistical functions and data comparison. Because of its ability to display observed and modeled data concurrently and perform basic statistical and data comparison functions, the postprocessor is a useful tool in model calibration and environmental systems analysis. The postprocessor window (Screen 10.14.3) consists of a tool bar, a plot area, a file information section, and a statistical functions section.

Screen 10.14.3

Loading and Viewing Data

The NPSM postprocessor has the ability to display three different data types—NPSM output files, BASINS water quality observation data, and USGS flow data.

When the NPSM postprocessor is executed through selection of a Pervious land unit, Impervious land unit, or a Reach from the "View Time Series Output" window, data are loaded automatically into the postprocessor. Although data are loaded, they cannot be viewed until a parameter is selected from the "Plot(s) Selection" window (Screen 10.14.4). Select a parameter from the pull-down menu. Click the right or left arrow to designate which axis will display the data scale. Click **OK** when finished. Data are then displayed in the plot area.

Screen 10.14.4

Additional data can be loaded into the postprocessor and viewed by using the tool bar buttons or by selecting the appropriate file type from the *File* menu heading. All data loaded into the postprocessor must have the same time interval—daily, monthly, or annual. Once data have been loaded, they cannot be viewed until the **Plots** button is clicked and a parameter is selected from the "Plot(s) Selection" window.

1. Load additional NPSM output by clicking the **NPSM Data** button. You will be prompted to select a file name. Recall that your NPSM output file is located in the directory that you designated in the "NPSM Output Manager", under the name automatically assigned to it. Generally, this location is the BASINS\MODELOUT\<project name> directory.

2. Load BASINS water quality observation data by clicking on the **BASINS Water Quality Data** button. You will be prompted to select a file. This is the file created when using the *Export Water Quality Observation Data* utility in BASINS. You can specify the location of the file during execution of the function; however, the default location is the BASINS\MODELOUT\OBS directory. The file will have an .obs extension. Refer to Section 7.4, Water Quality Observation Data Management Utilities, for directions on appending local water quality data to existing BASINS water quality observation data and exporting BASINS water quality observation data.

3. Load USGS flow data by clicking the **USGS Flow Data** button. You will be prompted to select a file.

   The BASINS USGS Gage Stations data layer contains mean flow, critical low flow (7Q10), and monthly mean flow data for some USGS gage stations. For a typical model calibration, however, it is suggested that you download daily flow data from the USGS's United States NWIS-W Data Retrieval web site. The URL is http://h2o-nwisw.er.usgs.gov/nwis-w/US/. From this web page, select the state of interest. On the subsequent page, either enter the USGS station number of interest and "Retrieve Data" or view a map, a list of counties, a list of basins, or a list of all stations to find the station of interest. After selecting the station of interest, observe the Data Types Available. For a typical model calibration, select the "Historical Streamflow Daily Values". Historical streamflow daily values are available for many but not all USGS stations. After selecting "Historical Streamflow Daily Values", enter the appropriate Dates to Retrieve and select the Output format as a "Tab-delimited text data file" with the "MM/DD/YYYY" date format. Retrieve these data and save them to your hard drive.

Postprocessor Components

1. The files and parameter names are listed in the file information window in the bottom right portion of the screen.

   To display a parameter or multiple parameters from a loaded file, click the "Plots" button located under the file information window. Select a file and parameter(s) from the pull-down menus on the "Plot(s) Selection" window (Screen 10.14.4). Once a parameter is selected, click the right or left arrow to designate which axis will display the data scale. Click **OK** when finished.

   The file information window lists the File Name, Color (as it's displayed in the plot area), Description, and six additional columns of information referring to statistical output (which will be discussed with the statistical functions). The scroll bar directly below the window lets you view all columns in the table. Loaded data can be removed from the postprocessor by highlighting the appropriate file name, clicking the right mouse button, and selecting Delete.

2. The "File Information" box above the window displays the range of dates for data loaded into the postprocessor. For NPSM output files, this period matches the simulation period assigned in the "Simulation Time and Meteorological Data" window. The BASINS water quality observation data period depends on the availability of data for the monitoring station(s) selected when executing the *Export Water Quality Observation Data* utility in BASINS. The period of record for the USGS flow data is specified when downloading the data from the USGS web site.

3. Clicking the "Plot Manager" button opens a plot manager or "Layer Control Data" window (Screen 10.14.5). This window is used to change the color, line type, and thickness of plots and/or to remove data previously loaded into the postprocessor.

   To remove data from the postprocessor, highlight the file name and click the **Remove** button. To remove all plots, click the **Remove All** button.

   To change a plot color, highlight the file name and double-click on the colored box next to "Color:". Select a new color from the "Color" window and click **OK**. Then click the **Update** button to save this change. Note that the colored box located next to the file name changes color.

To change a plot line type, highlight the file name and change the line type in the "Symbol" box. Click the **Update** button to save this change.

To change a plot or line thickness, highlight the file name and change the integer value in the "Line" box. Click the **Update** button to save this change. Note that the line number in the file table is updated. After making all necessary changes in this window, click the **Close** button.
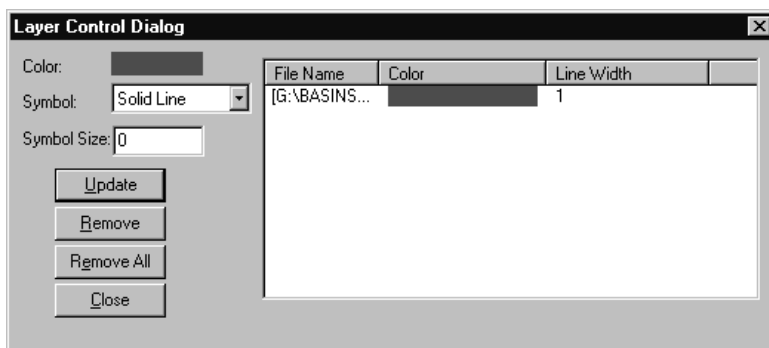
Screen 10.14.5

4. The plot area consists of the plot window, x- and y- axis labels and boundary designation boxes, and a "point display" section.

    The plot period initially displayed represents the entire extent of data loaded into the postprocessor.

    The time period represented on the plot can be changed by varying the Start and End dates using the boxes located under the x-axis. The plot period boxes represent month, day, and year (from left to right) for both the Start and End dates. Values in these boxes can be changed by clicking the cursor within a box and changing its value or by clicking on the up and down arrows next to each box. Depending on the time interval of the data loaded into the postprocessor, some of these boxes may not be active.

    The minimum and maximum value of the y-axis can also be changed in a similar manner.

    After making any changes to values in the plot period boxes, click the **Refresh** button located on the tool bar.

    The "point display" section of the graph is the area located between the two bold lines, containing the words "Selected" and "Corresponding value(s)". This region is used to display point values for the plot. Click the cursor at any location in the plot window. The date corresponding to the point where the cursor is clicked is displayed in addition to the associated parameter value. The parameter value is displayed in the same color as its plot. If multiple output files containing time series data for the same parameters have been loaded into the postprocessor, multiple parameter values appear in the "point display" section.

5. Statistical operations are executed from the "Statistical Calculation:" section of the postprocessor. The postprocessor calculates and plots either the running arithmetic or geometric mean for the parameters listed in the "File Information" section of the postprocessor (Screen 10.14.6). Statistical output

summaries are created and listed in the "File Information" section of the postprocessor. The statistical operation is performed for the time period shown in the plot window.

The statistical calculation can be modified by selecting either "Arithmetic Mean" or "Geometric Mean" from the "Statistical Method:" box and designating "Step" and "Threshold" values. The "Step" value refers to the incremental number of output file time steps over which the value is statistically averaged. For example, a step length of 30 for daily data would be used to plot the 30-day geometric mean.

The "Threshold" value is the value against which the parameter values are compared (typically a pollutant standard). The threshold value is displayed as a line on the statistical plot (y = threshold), and the statistical output summaries are based on this value. The "Step" and "Threshold" values are edited by entering a value directly into the appropriate boxes. Click on the **Refresh** button to plot the threshold value and recalculate the exceedance statistics.


Tip: To view a plot of the original data (not the statistical plot), set the "Step" to 1 and the "Threshold" to 0.

The statistical output summaries are presented in the "File Information" section for each file. The summaries contain the following information:

- No. of Exceedances—the total number of periods in which the running mean (either arithmetic or geometric) is above the threshold value. A single exceedance may consist of a single day or multiple consecutive days.

- Maximum No. of Days—the total number of days in the longest single exceedance, or the period over which the threshold is exceeded for the most consecutive days.

- Minimum No. of Days—the total number of days in the shortest single exceedance, or the period over which the threshold is exceeded for the least consecutive days.

- Total No. of Days—the total number of days during which the threshold value is exceeded.

- Exceedance Percentage—the total number of days during which the threshold value is exceeded, divided by the total number of days in the period of time represented on the statistical plot.

Negative and zero values in any data sets loaded are ignored during statistical calculations.

Tip: The statistical output summary table can be saved to a file by selecting the Save Report option from the File menu heading.
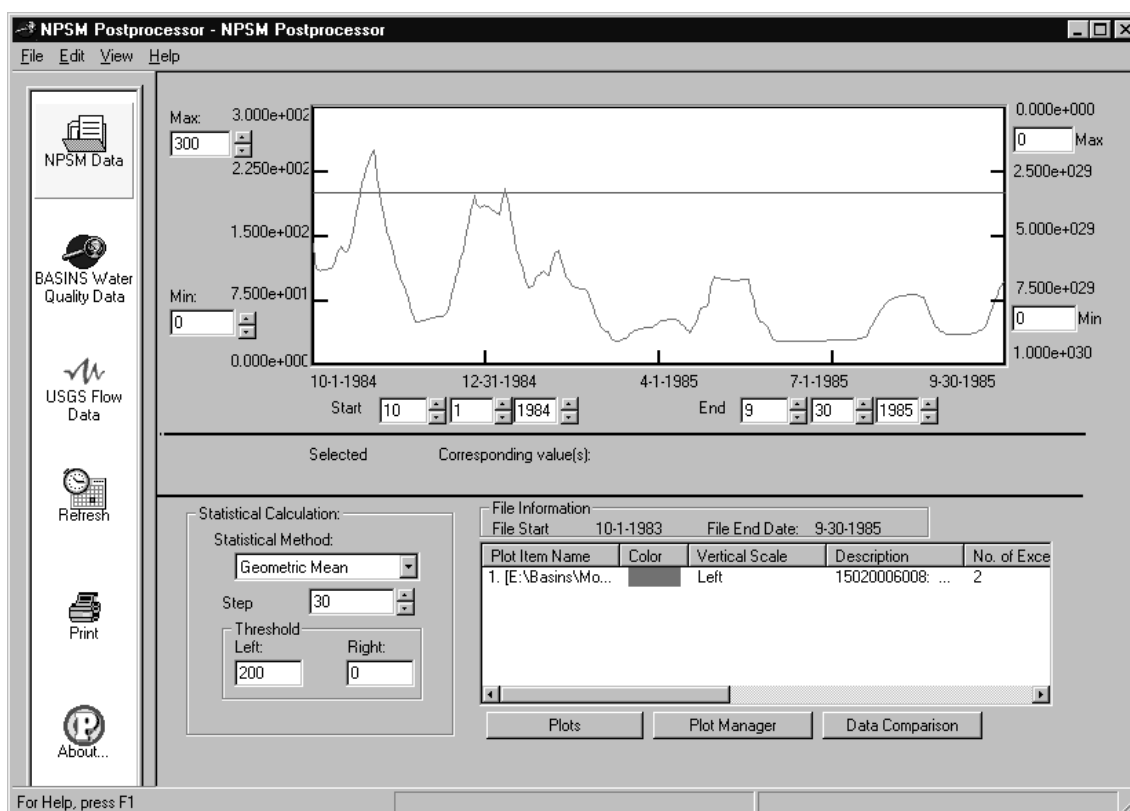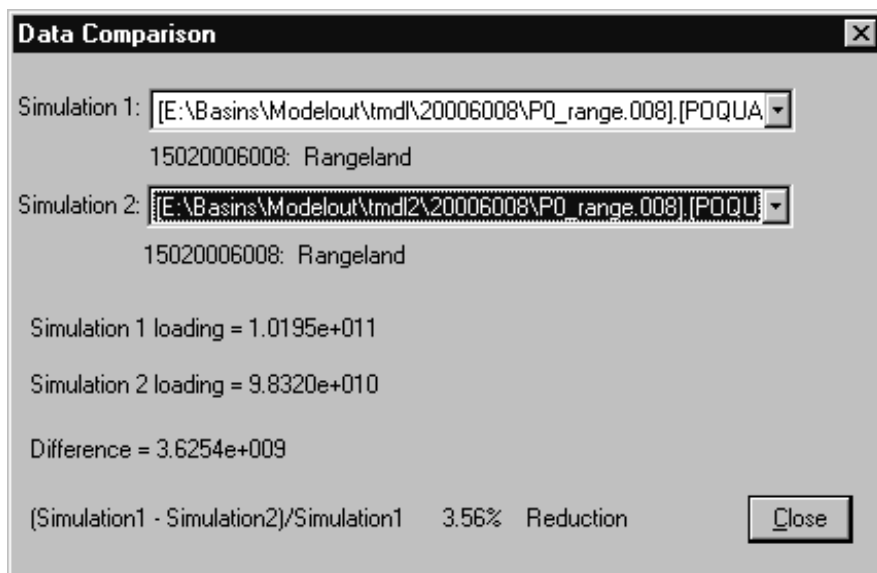

Screen 10.14.6

6. The postprocessor can also provide a basic comparison of data. This function is best applied to time series loading data from multiple simulation runs. It is executed by clicking the **Data Comparison** button.

   The "Data Comparison" window (Screen 10.14.7) prompts you to select two files—one in the "Simulation 1" box and another in the "Simulation 2" box. To compare different loading scenarios, the selected files should be from different simulation runs and should represent the same land unit, reach, or watershed. The resulting output displays:

   • The Simulation 1 loading value—the total loading integrated over the period plotted for the selected parameter in the "Simulation 1" file.

   • The Simulation 2 loading value—the total loading integrated over the period plotted for the selected parameter in the "Simulation 2" file.

   • The Difference—the difference between the Simulation 1 and Simulation 2 loading values (Simulation 1 - Simulation 2).

   • The % Increase or % Reduction—if the "Difference" value is negative, the corresponding % Increase value is displayed. If the "Difference" value is positive, the corresponding % Reduction is displayed.

Click **Close** to close the window. The loading values, or areas under each curve, are also listed in the file information window in the "Area Under the Curve" column.
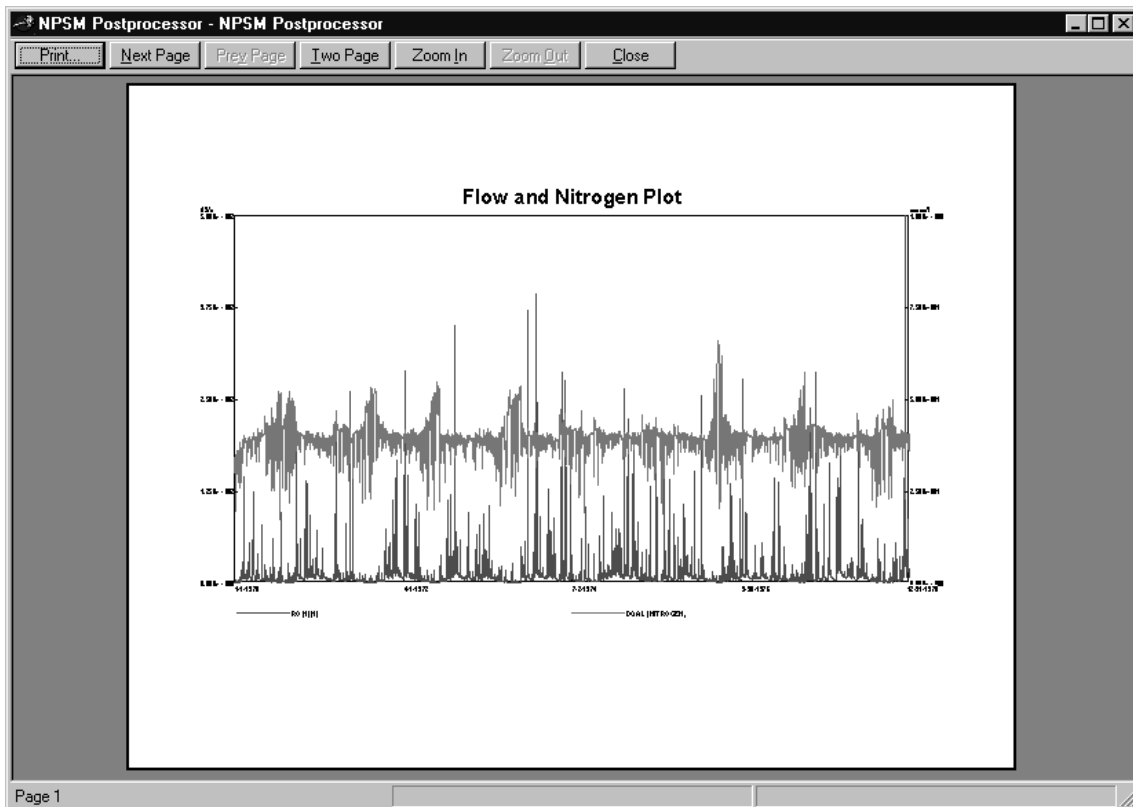

Screen 10.14.7

Printing Plots

The NPSM postprocessor can also print the plots displayed in the plot window. From the *File* menu heading, select *Print Preview* to view the page layout (Screen 10.14.8). From this window, you can print, view multiple pages, and zoom in or zoom out. To add a title to a plot, click **Close** to return to the NPSM postprocessor window, click on the *Edit* menu heading, and select *Title*.

The plot can also be printed by selecting *Print* from the *File* menu heading. Be sure that the proper printer settings have been made (select *Print Settings* from the *File* menu heading).

Screen Screen 10.14.8

TUTORIAL

- *In the "View Time Series Output" window, click the Reaches radio button.*
- *Highlight Blacklick Cr.*
- *Select Graph to execute the NPSM postprocessor. The data file you have loaded is the BASINS\MODELOUT\TUTORIAL\REACHESRO_BLACK.012 file. From the "Plot(s) Selection" window, select this file and the parameter DQAL (Fecal Coli.), by highlighting it in the parameter box and clicking the << button. Click **OK**. The file name and parameter will appear in the file information box.*
- *Change the x-axis time period to represent 1/1/1982 - 12/31/1982.*
- *Click on the plot area to view the fecal coliform concentration on any given day.*
- *Change the "Statistical Method" to Geometric Mean, the Step Length to 30, and the Threshold to 200. Click **Refresh**. Change the y-axis Max. to 400. The plot now displays a line at the 200 #/100 ml level and a plot of the 30-day geometric mean for fecal coliform for the year 1982. In the file information box, note the statistical information.*
- *In the Plot manager, remove the RO_Black.012 file. Click **Remove**. Click **Close**.*
- *Use the **NPSM Data** button to load the PO_agric.000 file in the BASINS\MODELOUT\TUTORIAL\ALLSHEDS directory. This file contains the total flow and total fecal coliform contributions from agricultural land in all subwatersheds.*

TUTORIAL *(cont)*

- *Use the **NPSM Data** button to load the PO_fores.OOO file in the BASINS\MODELOUT\TUTORIAL\ALLSHEDS directory. This file contains the total flow and total fecal coliform contributions from forest land in all subwatersheds.*

- *Click the **Plots** button and select the parameter SOQUAL for these two files.*

- *Change the x-axis time period to 1/1/1982 - 12/31/1982.*

- *Although the **Data Comparison** button is typically used to compare one simulation run to another, it can also be used to compare pollutant loadings from different land uses. Click the **Data Comparison** button. In the Simulation 1 box, select PO_agric.OOO. In the Simulation 2 box, select PO_fores.OOO. Note the total fecal coliform loadings from each land use (in the Simulation 1 loading and Simulation 2 loading boxes), the difference between the two loadings, and the reduction (which in this example represents the percent difference between loadings from the two land uses).*

- *Exit the NPSM postprocessor to return to the "View Time Series Output" Window.*

10.15 Creating an NPSM Default File

It is highly recommended that all model parameter values be carefully considered and selected prior to running NPSM. While the parameter values in the starter.def file included with the BASINS system will permit the user to run the model, the values may not be representative of your region or watershed. This section outlines the steps required to develop a custom default file from scratch or to modify an existing default file.

1. Run NPSM as a stand-alone program, as described in Section 10.2. Or, if you are currently in the interface, be sure that no NPSM project is open. If a project is open, select *Close* from the *Project* menu heading.

2. From the *Default* menu heading, select *New*. Notice that only the following NPSM options become active during default file development: **Save the current project**, **Reach Editor**, **Land Use Editor**, **NPSM Control Cards**, **Pollutant Selection Screen**, and **Input Data Editor**.

3. In the **Reach Editor**, click the **Add/Remove Reaches** button. Add a reach by clicking the right mouse button while the cursor is in the table and selecting the appropriate option. Designate a Reach #, a Reach Name, and the # of Exits. The Watershed column does not need a value. Click **OK** to save the designations and continue. Click **Done** to exit the "Reach Editor" window.

4. In the **Land Use Editor**, add a land use by clicking the right mouse button while the cursor is in the table and selecting the appropriate option. Designate the land use name in the Land Name cell. If your text does not fit in the allocated space, click on the bar between the Land Name and Land Type column headings and extend the right side of the column to the right. Designate the land use as pervious or impervious. The radio buttons in the Land Type box at the top of the screen can be used to change the designation. The Area and Watershed columns do not need values. Add additional pervious and impervious land uses as necessary. Click **OK** to save the designations and continue.

5. In **NPSM Control Cards,** specify all HSPF modules for which you will declare default values. It is a good idea to select all modules you expect to use, even if you currently do not plan to populate the database for certain modules. Note that "Impervious Land" will not appear in the list if no impervious land uses were defined and "Pervious Land" will not appear if no pervious land uses were defined. Click **Close** to exit the "NPSM Control Cards" window.

6. In the **Pollutant Selection Screen**, specify all pollutants you intend to parameterize for modeling. Once again, it is a good idea to select all pollutants you expect to use, even if you do not plan to populate the database in its entirety. Click **OK** to save the designations and continue.

7. The most important step in creating a default file is designation of module settings and parameter values in the **Input Data Editor**. Be sure to populate all module settings and parameter values required for model execution in the PERLND (pervious land), IMPLND (impervious land), and RCHRES (reaches) groups. After defining all module settings and parameter values associated with each land use, reach, and pollutant, click **Done** to save settings.

> *Tip:*   *Refer to the "Input Data Editor" portion of the NPSM discussion (Section 10.11) for more specific information related to entering and editing data.*

> *Tip:*   *Appendix B of this manual contains an HSPF data dictionary containing all HSPF parameter names, definitions, units, and minimum and maximum acceptable values. This is a valuable resource for populating the "Input Data Editor".*

8. Click on the **Save the current project** button to save your default file. Be sure that the default file is assigned a .def extension. Usually, default files are saved in the BASINS\MODELS\NPSM directory.

> *Tip:*   *You can also view and/or modify an existing NPSM default file by selecting Open from the Default menu heading.*